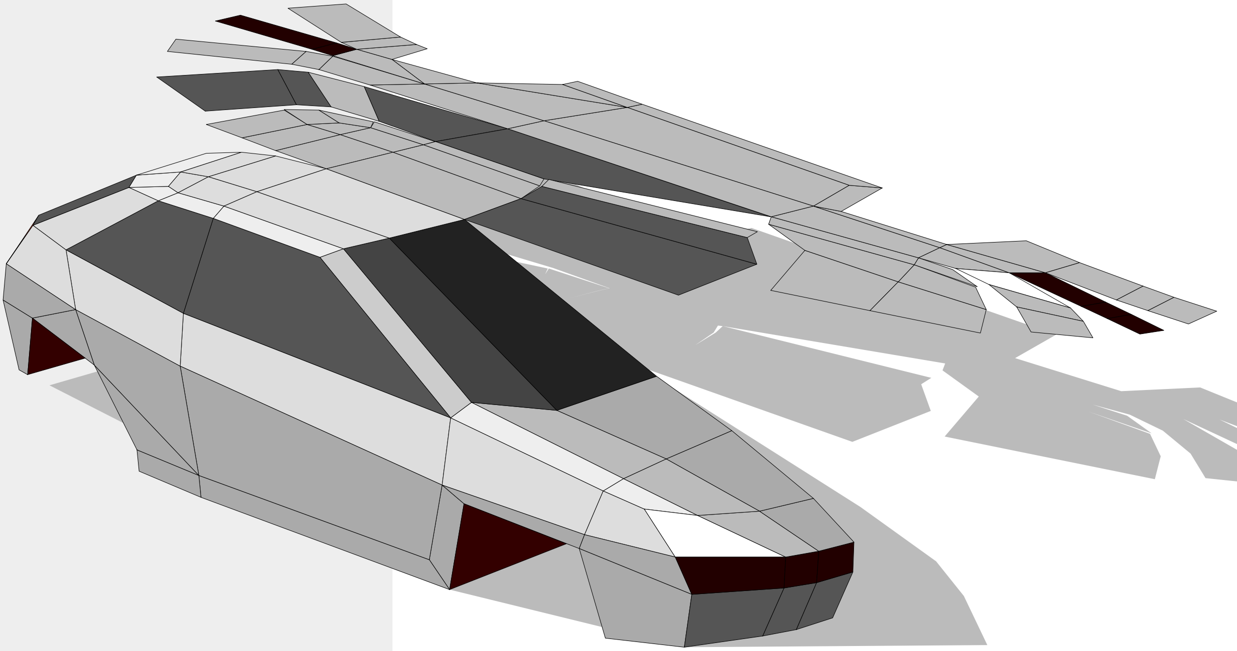
TouchCAD

Version 3.5

User's Manual



Limited warranty and disclaimers

A. Limited Warranty on Media

Lundström Design warrants the disks on which the Software is recorded to be free of defects in materials and faulty workmanship under normal use for a period of ninety (90) days from the date of delivery as evidenced by a copy of the sales receipt. Lundström Design will, as its option, replace or refund the purchase price of the diskette at no charge to you, provided you return the faulty diskette with the sales receipt to Lundström Design, or an authorized Lundström Design Dealer. Lundström Design shall have no responsibility to replace or refund the purchase price of a diskette damaged by accident, abuse, or misapplication. ANY IMPLIED WARRANTIES ON THE DISK, INCLUDING WARRANTIES OF MERCHANT ABILITY AND FITNESS FOR A PARTICULAR PURPOSE, ARE LIMITED IN DURATION TO 90 DAYS FROM THE DATE OF DELIVERY.

B. Disclaimer of Warranty on Software

THE SOFTWARE IS PROVIDED ”AS IS” WITHOUT WARRANTY OF ANY KIND, AND LUNDSTRÖM DESIGN EXPRESSLY DISCLAIMS ALL IMPLIED WARRANTIES, INCLUDING BUT NOT LIMITED TO THE IMPLIED WARRANTIES OF MERCHANT ABILITY AND FITNESS FOR A PARTICULAR PURPOSE. LUNDSTRÖM DESIGN DOES NOT WARRANT, GUARANTEE, OR MAKE ANY REPRESENTATIONS REGARDING THE USE OR THE RESULTS OF THE USE OF SOFTWARE OR ANY ACCOMPANYING WRITTEN MATERIALS IN TERMS OF THEIR CORRECTNESS, ACCURACY, RELIABILITY, CURRENTS, OR OTHERWISE. YOU ASSUME THE ENTIRE RISK AS TO RESULTS AND PERFORMANCE OF THE SOFTWARE AND WRITTEN MATERIALS. LUNDSTRÖM DESIGN DOES NOT WARRANT THAT THE SOFTWARE WILL WORK CORRECTLY IN YOUR MULTI-USER OR NETWORK ENVIRONMENT. IF THE SOFTWARE IS DEFECTIVE, YOU, AND NOT LUNDSTRÖM DESIGN OR ITS DEALER, DISTRIBUTORS, AGENTS, OR EMPLOYEES, ASSUME THE ENTIRE COST OF ALL NECESSARY SERVICING, REPAIR, OR CORRECTION. SOME STATES DO NOT ALLOW THE EXCLUSION OF IMPLIED WARRANTIES, SO THE ABOVE EXCLUSION MAY NOT APPLY TO YOU.

C. Complete Statement of Warranty

THE LIMITED WARRANTIES PROVIDED IN PARAGRAPHS ”A” AND ”B” ABOVE ARE THE ONLY WARRANTIES OF ANY KIND THAT ARE MADE BY LUNDSTRÖM DESIGN ON THIS LUNDSTRÖM DESIGN PRODUCT. NO ORAL OR WRITTEN INFORMATION OR ADVICE GIVEN BY LUNDSTRÖM DESIGN, ITS DEALERS, DISTRIBUTORS, AGENTS, OR EMPLOYEES SHALL CREATE A WARRANTY OR IN ANY WAY INCREASE THE SCOPE OF THIS WARRANTY, AND YOU MAY NOT RELY ON ANY SUCH INFORMATION OR ADVICE. THIS WARRANTY GIVES YOU SPECIFIC LEGAL RIGHTS. YOU MAY HAVE OTHER RIGHTS, WHICH VARY FROM STATE TO STATE.

D. Limitation of Liability.

IN NO EVENT WILL LUNDSTRÖM DESIGN, OR ITS DEVELOPERS, DIRECTORS, OFFICERS, EMPLOYEES, OR AFFILIATES BE LIABLE TO YOU FOR ANY CONSEQUENTIAL, INCIDENTAL, OR INDIRECT DAMAGES (INCLUDING DAMAGES FOR LOSS OF BUSINESS PROFITS, BUSINESS INTERRUPTION, LOSS OF BUSINESS INFORMATION, AND THE LIKE) ARISING OUT OF THE USE OF OR INABILITY TO USE THE SOFTWARE OR ACCOMPANYING WRITTEN MATERIALS, EVEN IF LUNDSTRÖM DESIGN OR AN AUTHORIZED LUNDSTRÖM DESIGN REPRESENTATIVE HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES.

BECAUSE SOME STATES DO NOT ALLOW THE EXCLUSION OR LIMITATION OF LIABILITY FOR CONSEQUENTIAL OR INCIDENTAL DAMAGES, THE ABOVE LIMITATIONS MAY NOT APPLY TO YOU.

Lundström Design liability to you for actual damages for any cause whatsoever, and regardless of the form of action, will be limited to the greater $200 or the money paid for the Software that caused the damages.

Introduction

TouchCAD 3.5 is a combined 3D-modeling and unfolding program. It can be used for creating and building complex free-form 3D-shapes such as sheet metal objects, tents, sail roofs, blimps, boat hulls, architectural and industrial design models, reversed engineering to recreate copies of already existing objects, etc. The ability to unfold just about anything in a controlled way enables you to design and build shapes that you would normally hesitate to design because of the difficulties of extracting real usable drawings and assembly instructions.

Lundström Design
Ekhagsvägen 7
SE-104 05 Stockholm
Sweden
Tel: int+46-8-15 46 63

info@touchcad.com
www.touchcad.com

Table of contents

Introduction

Palettes

Tool palette

Menus

File menu

Edit menu

Page menu

Tool menu

View menu

Palette menu

Contextual menu

Quick keys and Shortcuts

Unfold view

Tutorial examples

Boat hull

A human shape

Appendixes

Importing files

Index

Installing TouchCAD

Notes and Acknowledgements

Installing TouchCAD

Move the document called TouchCAD and the sample files to a suitable location on the hard disk. Double-click on the symbol to decompress the files (using StuffIt Expander on Mac and WinZip on Windows). On MacOS Classic you need to have QuickDraw3D installed. It is included in QuickTime, which you can find on the System CD.

If you run on MacOS X or Windows 32, install Quesa. The latest version of Quesa can be found at www.quesa.org.

TouchCAD requires QuickTime to run. Most Mac users already have it in the System and do not need to install it, unless you want to upgrade to the latest version. Windows users can download the latest version of QuickTime for Windows from Apple at www.apple.com.

Read and install the copy protection instruction found on the program CD.

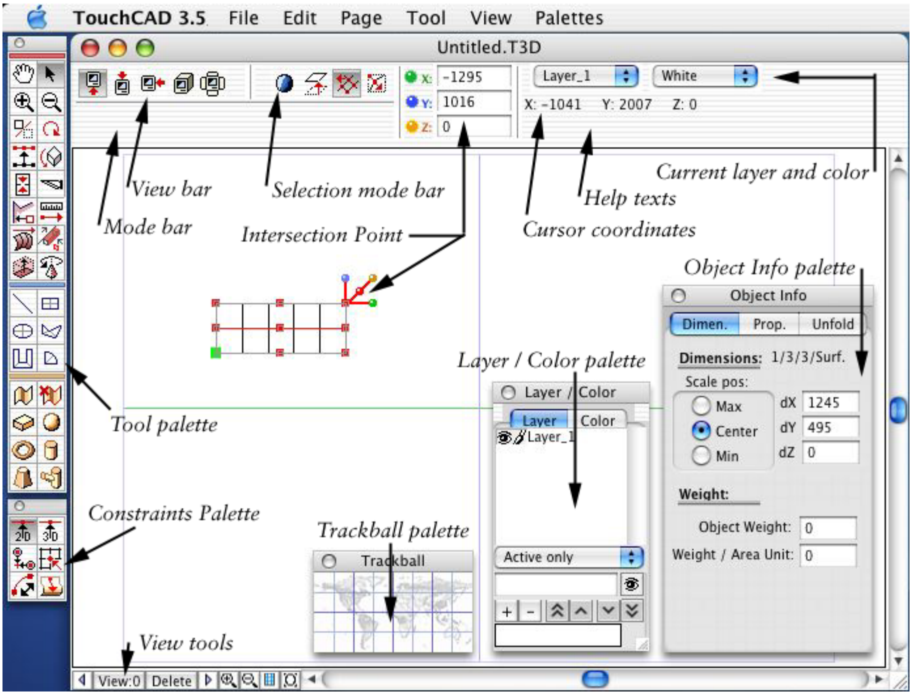
Hardware configuration

TouchCAD will work on any PowerMac equipped with System 8.6 or later (MacOS X included). A G3 processor or better is recommended. TouchCAD is also available for Windows 32 based systems where a 300 MHz Pentium or better is recommended. Windows 32 also requires OpenGL and Quesa to work. On all versions we recommend an accelerated video card to get access to all rendering effects.

Video cards.

It may be a good idea to test the speed of the video card to find the best possible performance. Computers with no video acceleration typically work best using 256 colors. Older video cards often work best using thousands of colors, whereas newer video cards work best using millions of colors. Note that you need to restart the program after having changed color depth to get correct results.

Starting the program



This is roughly what you see when you start the program though it varies slightly due to the operating system used. The ***Tool Palette*** contains tools for drawing, selecting and modifying objects, and basic navigation tools such as zoom, pan, etc. The palette is sub-divided into three parts, from the top, tools for editing (the red section), 2D-shapes (the blue section), and 3D-shapes (the yellow section). Some tools in the ***Tool*** palette contain options showing up in the ***Mode bar*** when a tool is selected. The ***Layer / Color*** palette is used for adding, deleting, activating and editing of layer colors. The Object Info palette is used for changing various properties of the drawing objects. The ***Trackball*** palette is used for rotating/paning and the display view. The ***View*** bar contains editing ***views (Front, Top, Side/Right, Unfold and Perspective***). The ***Selection Mode*** bar is used for choosing a selection mode when the ***Selection Arrow*** tool is activated. The options from left are: Select everything in the current layer, select objects as elements, select all vertexes connected to the ***control point knot*** you click on, and select ***one control point*** at a time. When something is selected, a red marker called the ***Intersection Point*** is displayed in the drawing area. The ***x, y, z coordinates*** of this point are displayed above the drawing area. Dragging, entering new coordinates in the edit fields or using the ***arrow keys*** (+/- for moving away/towards you) can be used for moving the location of the ***Intersection Point***. The cursor coordinates, help text, and layer/color info are also shown above the drawing window.

Some basic concepts

The main drawing window consists of some basic elements:

The drawing area

The drawing area is where you draw and edit the model. It displays a gray rectangle and a horizontal line and vertical line when starting the program. The rectangle indicates where the printable area is. It is perfectly OK to place objects outside this rectangle, but they will not be printed, so in most cases it is practical to stay inside it. The rectangle does not reflect the actual ***sheet*** size, such as A4 or Letter. Instead it shows the actual printing area you have access to, as most printers can't print to the edge of the ***sheet***. The advantage with this method is that TouchCAD allows you to print over just about any number of sheets of ***sheet***. You can for example print out an A3 drawing on an A4 printer by doing it on two sheets of ***sheet***. Just cut one of the sheets along the edge where the lines stop, and place it over the adjacent sheet.

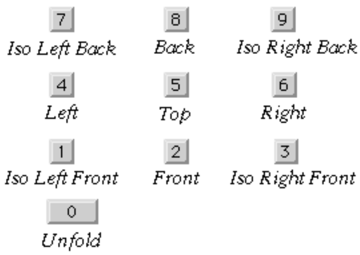
The horizontal and vertical ***lines*** displayed in the Front, Top and Side (right) views represent the location of the ***zero point coordinates*** in the respective directions. The zero point is normally located in the middle of the drawing, but you can move it using the Move Print Area command in the Page menu.

The View Bar

The ***View Bar*** contains the standard view buttons, ***Front*** *(1)*, ***Top*** *(2)*, ***Side*** *(3)* (right), ***Unfold*** *(4)* (the unfolding feature is regarded as a view in TouchCAD) and ***Perspective*** *(5)*. You can also change view by using the standard view settings in the ***View*** menu, by pressing the ***0-9 keys*** on the numerical keyboard (see the picture below), by using the ***Rotate View tool*** in the ***Tool*** palette, or by using the ***Trackball*** palette.

To take a look at a model in full 3D, hold the cursor over the ***Trackball*** palette push the mouse button and drag. The model now rotates. Note that you rotate the view, not the model.

You can reset the rotation by clicking on either of the ***view*** buttons in the ***View*** bar. Other ways to reset views is to select ***Standard Views*** in the ***View*** menu, or by using the numerical keyboard indicated below with views.



Auto centering when changing view

TouchCAD provides an ***auto-centering*** feature automatically ***centering*** the view on the ***midpoint of the selected object(s)***. If ***nothing*** is ***selected***, it centers on the ***mid-point*** of the ***entire visible model***. When used in the intended way it typically allows you to continue working directly after having changed view. It is therefore good practice to consider what you have selected before changing the view.

The Mode Bar

The *Mode Bar* located to the left and immediately above the drawing area sometimes displays various ***sub options***. The options vary from tool to tool and some tools in the Tool palette do not have any options at all. If you for example select the ***Rectangle*** tool in the ***Tool*** palette *(1)* in the Blue 2D section of the tool palette) you will find four sub-options in the ***Mode*** bar *(2-6)*. The first three control the shape type, from the left; a rectangle defined as a ***curve*** (a series of ruler points) *(2)*, a rectangle defined as a ***ruler*** *(3)*, and the last a rectangle defined as a ***surface*** *(4)*. The last button defines the ***depth*** of the object (in a direction away/towards you) *(5)*. If the depth is zero you get a flat object, if not a 3D cube is obtained. The current depth setting is displayed to the far right *(6)* in the ***Mode*** bar.

The Selection Mode bar

The ***Selection*** mode bar is used for controlling how objects are being selected using the ***Selection Arrow*** tool. The ***Global*** mode *(1)* selects ***all objects*** in the active layer. The ***Object*** mode *(2)* selects and edits individual ***objects***. The ***Mesh*** mode *(3)* selects and edits individual ***control points*** and ***control point knots***. ***Mesh*** mode selects ***all points*** in a ***control point knot*** if you click on it. The ***Vertex*** mode *(4)* selects ***individual control points***, one at a time. If you click on a control point knot, it only selects ***one control of the points*** in that knot.

The Intersection coordinate fields.

The ***Intersection Coordinate*** fields represent the ***Intersection Point X, Y and Z coordinates***. The ***Intersection Point*** symbol, as seen to the left, is similar to the intersection point in a word processor and essentially tells you where you are in the model. The color dots at the end of the symbol axes are used for dragging the selection in various ways and directions. The bottom left dot is for freeform dragging. The blue, yellow and green dots are used for constrained dragging in a fixed direction. The intermediate red dot is used for moving the selection away/towards you to the closest non-selected point.

The Intersection Point coordinates also act as working planes when drawing new objects. If you, for example, draw a circle in the Top view, it will be placed at the Y location of the Y field.

These fields have ***three*** modes. In the first mode you simply enter a new coordinate, and the location of the selection is updated accordingly. Other selected objects will move the same distance as the Intersection point.

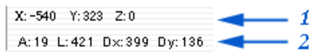
In the ***second*** mode you place an ***"="*** character before the number and that provides all selected objects with the same ***absolute coordinate value*** (e.g. =100 means that all objects will get the coordinate value 100).

In the ***third mode*** you insert an ***"M"*** before the coordinate and that means that you want to move the object ***relative to the previous coordinate*** (e. g. if the previous coordinate was 500 and you type in M100 you will get 500 + 100 = 600).

The Layer and Color pop-ups

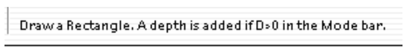
The ***Layer and Color pop-ups*** located above the drawing area are used for ***setting the active/default layer/color***. Choose for example Blue in the color pop-up, and all new objects that you draw will become blue by default. The ***Layer / Color palette*** also allows you to change the ***active/default layer/color***.

Note that the active / default setting is not the same as the setting of an object. You change the ***object layer*** and ***color properties*** by selecting the object(s) and then modify it in the ***Object Info palette***.



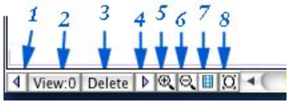
The cursor coordinates

The ***cursor coordinates*** are displayed above the drawing window *(1)*. In some cases the ***Info*** line is used for displaying ***additional data*** *(2)*. This primarily happens when you draw new objects. In the example above it shows the angle between the previous point and the current cursor location, the diagonal (polar) length, and the offsets in the horizontal and vertical directions.



The Info Line.

The ***Info*** line above the drawing window displays various instructional messages.



The Scroll Bars

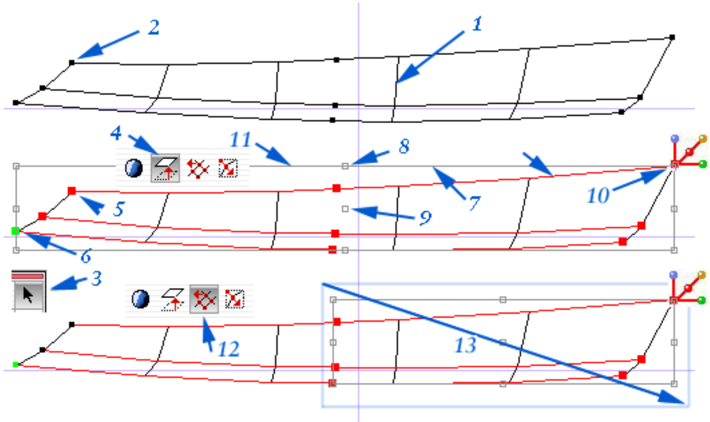
The ***Scroll*** bars allow you to scroll the drawing area horizontally and vertically in a fairly standard way used by most other programs.

In the bottom left corner of the drawing window you will find seven buttons. The ***first four*** are used for ***saving*** *(2)*, ***deleting*** *(3)* and ***toggling between views*** *(1, 4)*. You can ***save*** up to ***ten views*** in the ***wireframe*** mode and up to ***ten views*** in the ***render*** mode. The wireframe view saving settings include the ***view***, ***degree of zoom and pan***. The ***render mode*** settings ***include all settings*** found in the ***Mode*** bar while you are in the ***render*** mode. ***Saved views*** are ***stored*** as a ***part of the drawing*** and can be re-used when you open a drawing. The ***left*** *(1)* and ***right*** *(4)* arrows are used for stepping between saved views. Clicking on the ***Save*** *(2)* button ***saves*** the current view. Clicking on the ***Delete*** *(3)* button ***deletes*** the current view from the list.

The ***following two buttons*** with the ***zoom in*** *(5)* ***/ out*** *(6)* symbols ***double*** and ***halves*** the degree of ***zoom***, essentially as if you double-click on the Zoom tools in the Tool palette.

The ***blue button*** *(7)* adjusts the degree of ***zoom and pans*** so that the ***sheet*** area fits into the drawing window.

The ***last button*** *(8)* is used for ***centering*** the view on the currently ***selected object***.



Open a document called Manual1.T3D. It contains a model used for explaining some basic elements of TouchCAD. The picture below shows how it looks.

Control Points

When you first open the model it will look like the upper picture (*1*). The black dots are called ***Control points*** (*2*). They are shown black because they are not selected.

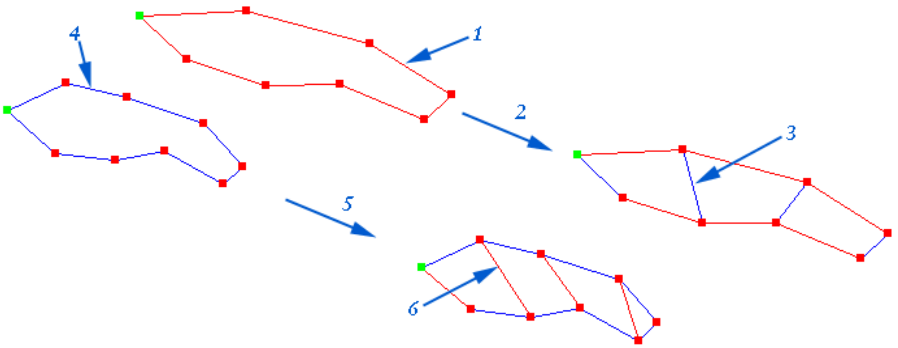
Choose the ***Selection Arrow*** *(3)* tool in the ***Tool*** palette. Click on the ***Object*** selection (*4*) mode in the ***Selection*** mode bar (*4*). Click on one of the black ***Control point*** (*2* ***or*** *10*) dots to select the object. All black dots become ***red*** (*5*) and slightly bigger than before, indicating that the entire object has been ***selected***. The ***Object*** selection mode selects ***objects as a unit*** explaining why all points are selected. The ***green dot*** represents the ***starting point*** (*6*) and is an important element when editing objects. The ***green dot*** is a ***control point*** as well. It is black if nothing is selected and green if something is selected to indicate where the starting point is located. The green dot increases in size when the point is selected. One of the control points holds the ***Intersection Point*** (*10*) symbol used as a handle for moving the selection in space.

The ***Mesh*** *(12)* selection mode selects individual control points. Several points can be selected by drawing a ***selection rectangle*** *(13)* using the ***Selection*** arrow *(3)* tool.

A gray ***selection box*** *(7)* also occurs when something is selected. The ***box handles*** *(8-9)* allow you to re-scale the selection from the corners, from the line midpoints in one direction or from the ***middle*** (*9*). The corner selection handles scales proportionally when the ***Shift key*** is pressed. The ***center-point*** (*0*) scales in one direction only if the Shift key is pressed and when dragging horizontally or vertically, and proportionally when dragging diagonally. Other selection box options include ***skewing*** press *Opt/Alt* while selecting the box ***midpoints***, ***deforming*** by pressing the *Opt/Alt* while selecting a ***corner point***, and ***rotation*** by clicking on ***the lines in between the control points*** *(11)* and then rotate. When ***pressing*** the *Alt/Opt* key it rotates around the ***diagonal corner*** of the box, if not it rotates around the ***midpoint***.

Control Rulers

Some mesh lines become ***red*** when you select at least one control point in an object. These ***red lines*** are called ***Control Rulers*** (***2,6 below***) and are used for showing the ***object direction*** and the ***ruler identity*** of individual ***control points***. All control ***points along*** a given control ***ruler*** belong to the ***same control ruler***. Both the control rulers and the control points are ***indexed from*** the ***green starting point dot***.



Visual difference between curves, rulers and surfaces

Curves (a series of ruler points) and *control rulers* are *not surface objects* and are therefore not visible when rendered and can not be used for unfolding.

In the first example we have a *control ruler (1)*. You can see that it is a *ruler* because *all lines are red between the control points* even though this particular object is blue. You can see the color when the object isn't selected. The basic shape can be converted into a *surface (3)* by using the *Curve to Surface* (*2*) command found in the *Tool* menu. Note that you can see that it is a *surface* because it *has two control rulers* and *blue intermediate lines* (the color of the object).

In the second example we have a *curve (4) (a series of ruler starting points)*. You can see that it is a *curve* because *the lines are not red between the control points* even though it is selected. It can be converted into a *surface (6)* using the *Curve to Surface (5)* command in the *Tool* menu. Note that you can see that it is a *surface* because it has *two red control rulers* and *blue intermediate lines* (the color of the object).

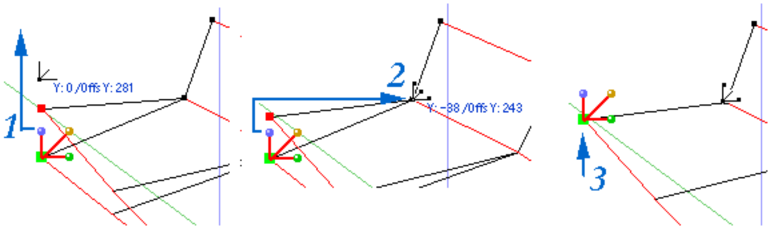
The *Object Info palette* also specifies the *data type* of the object holding the Intersection Point.

The Intersection point

One of the control points has a *red symbol (left)* that looks a bit like a coordinate system symbol. This point is called the *Intersection Point* and works a bit like the intersection point found in most word processing programs, where it usually is a vertical blinking line.

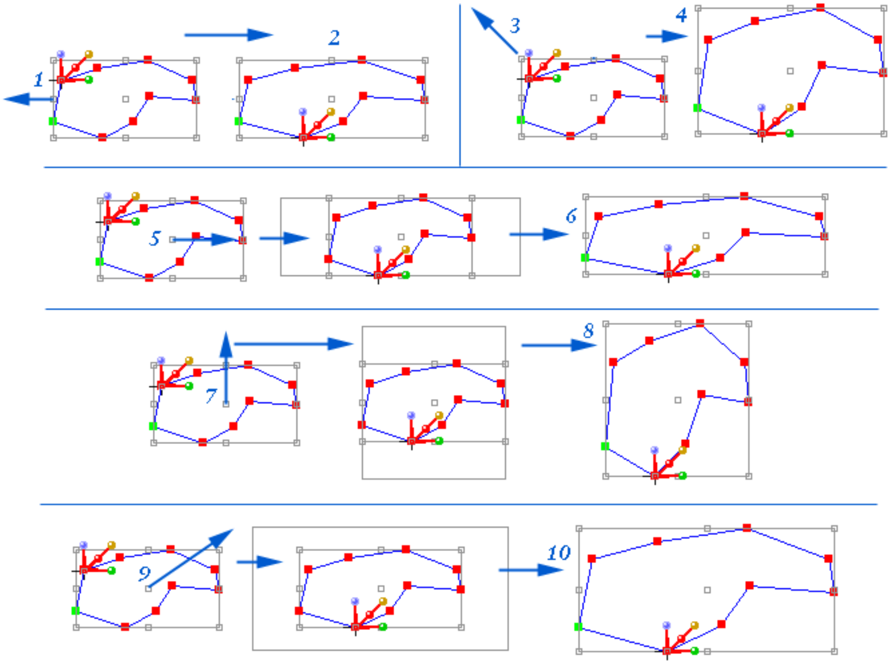
The purpose of the intersection point in a word processor is to indicate where letters will occur when you press a key on the keyboard. In TouchCAD the *Intersection Point* indicates where you are in the model, and you can read and change the coordinates of that particular location in ***the X, Y and Z coordinate (right)*** edit fields above the drawing area. The ***Intersection point*** moves to a new location when you click on another control point, and that is exactly what you would expect in a word processor.

The Intersection Point symbol contains several dots in different colors, and a corresponding set of dots can be found in front of the Coordinate fields above the drawing area. The red dot, found in the bottom left corner, is used for unconstrained dragging. The blue, green and yellow dots are hotspots for constrained dragging, and the direction is indicated by a dot in front of the respective coordinate fields above the drawing area. Note that you can actually drag in any direction with these three points, even away, towards you, and from any 3D-view. When dragging away/towards you really can't see what you are doing, but it can however be seen numerically as a floating text while dragging, specifying the actual coordinate as well as the offset from the starting point.



A very useful feature with the ***Constrained dragging mode (1)*** is that it is possible to ***refer to another control point*** *(2)* or mesh point and pick up its coordinate by simply holding the cursor over the given point. A snap cursor occurs and the selection jumps to the given position. The Intersection Point moves to the coordinate of the referred point (*3*) and in the direction used when clicking on it.

In the fixed Front, Top and Side views, an ***additional red dot*** occurs in the middle of the diagonal line of the Intersection Point symbol. This point is used for ***moving*** the ***Intersection Point*** to the closest control or mesh point found within the snapping area, in the direction ***away/towards*** you.. If there are several possible candidates, it will choose the closest point measured from the original location of the Intersection Point. If no points are found, if will not move at all.



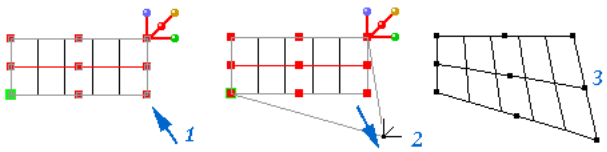
The Resize box

A ***gray rectangle***, called ***the Resize box*** (1), occurs around the selection. The rectangle is used for ***resizing*** the selected ***objects*** in various directions. Nine ***resize box handles*** appears ***around the edge*** of the ***resize*** box and ***one in the middle*** (5). You can start resizing when the diagonal ***Resize arrow cursor*** occurs, and by ***clicking*** on one of ***resize box handles***. Drag and then click again to finish the sequence.

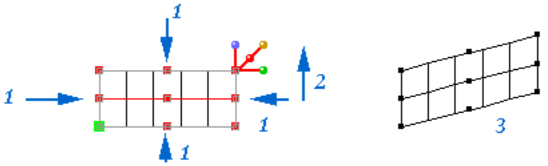
The four ***corner-handles (3)*** allows both ***horizontal*** and ***vertical scaling*** and ***proportional scaling*** if you press the ***Shift*** key.

The four ***midpoint handles*** allow constrained scaling ***horizontally*** and ***vertically***.

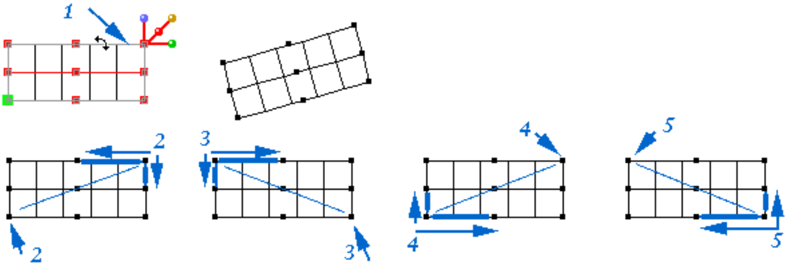
The ***center-point*** is used for ***scaling*** from the ***center (5)***. Drag to the ***right*** to make it ***wider*** (5), ***left*** to make it ***narrower***, ***up*** to make it ***higher*** (7), ***down*** to makes it ***lower***. You can choose to just ***scale*** in ***one direction*** by pressing the ***Shift key*** and drag either ***horizontally (5)*** or ***vertically (7)***. You can ***scale*** it ***proportionally*** from the ***center*** by dragging ***diagonally (9)*** while pressing the ***Shift*** key.



Holding the cursor over one of the corner points of the Selection box (1) can deform the selection. Press the ***Alt/Opt*** key ***click and drag*** *(2)*. The ***result*** *(3)* .can be seen to the right.

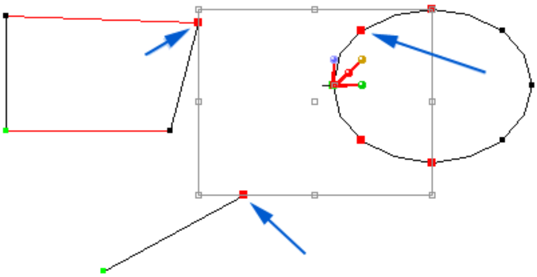


A similar effect can be achieved by using the ***midpoints*** of the ***Selection Box*** *(1)*. Press the ***Alt/Opt*** key ***click and drag*** *(2)*. The ***result*** *(3)* .can be seen to the right.



The ***lines (1) in between*** the control points of the selection box are used for ***rotation*** of the selected objects around the midpoint of the selection. Click and drag when the ***Rotation cursor*** is shown (***1***).

Holding over for example the ***top right segment*** *(2)*, ***Alt/Opt-clicking*** and dragging provide an alternative rotation point. It then rotates around the diagonal point in this case the bottom left corner. Consequently, the top left segment (***3***) rotates around the bottom right corner, the bottom left segment (***4***) around the upper right corner and the bottom right segment (***5***) around the upper left corner.



The ***resize*** box is a rectangle that encloses the selection, regardless of whether all control points are selected or not. The picture shows an example where some control points in many objects have been selected, using the ***Selection Arrow*** tool, the ***Mesh*** selection mode and by drawing a ***selection box***. All control points within the box are selected and the ***Resize*** box encloses only parts of the surface.

Note that ***control points normally override*** the ***resize box*** if it is located over ***resize box control handle***. You can however get access to all resize points by turning off the snapping features in the ***Constraints*** palette.

The ***resize box*** feature can be turned on and off at any time using the ***Resize Box*** command found in the ***View*** menu, or by using ***the Command (M) / Control (W) + "<" quick-key*** feature.

Moving objects

Objects can be ***moved*** by ***dragging***, by using the ***arrow*** keys, by ***entering new coordinates*** in the edit fields above the drawing window, or by using the available editing tools such as the rotate and resize tools etc.

Choose the ***Selection Arrow*** from the ***Tool*** palette, and activate the ***Object selection*** mode in the ***Mode*** bar. ***Select*** an object ***by clicking*** on it. ***Click again*** on one of the red dots to start the dragging procedure. The surface now follows the cursor. Note that you don't have to press the mouse button while dragging. ***Click*** again to ***stop*** moving.

To move ***one control point***, firstly click beside the object in the white drawing area to deselect the object. Then click on one of the points using the ***Selection Arrow tool*** in the ***Tool*** palette and the ***Mesh selection*** mode in the ***Selection mode*** bar. The red ***Intersection Point*** indicator appears again indicating that the point is selected. ***Click*** again to ***start*** the moving sequence. ***Click*** again to ***stop*** at the new location. Note that the surface changes shape when you drag. Try moving some other control points.

You can ***Shift-Click to activate further points***.

You can also draw a ***selection rectangle***, where every control point inside will be selected. Note that if you use the ***Object*** selection mode make sure ***that all control points*** in the object are really ***enclosed*** by the ***selection rectangle*** to ***select an object***. If not, the object will not be selected.

Using the Arrow keys

The ***Arrow*** keys move all selected objects one grid step in the respective direction and away/towards you when you use the plus and minus keys. Note that the movement affects different coordinates in the Front-, Top-, Side-, Perspective- and Unfold- views. The Up/Down keys, for example, increase and decrease the Y coordinate in the Front and Side views and the Z coordinate in the Top view. The Perspective view behaves as in the Front view when using the arrow keys. The same applies to the Unfold view when the Edit 3D mode is activated. More on that in the chapter about the Unfold view.

Nudge pan, zoom, and rotate view.

The arrow keys are also used for quick nudge panning when the Shift key is pressed, nudge rotating the view by pressing the Opt/Alt key, and for nudge zooming by pressing the Command (on Mac) / Windows (on Windows) keys.

The Selection mode bar

The ***Selection mode bar*** is used for controlling the selection and editing of objects when using the ***Selection Arrow tool*** in the ***Tool palette***. The options are from the left: The ***Global mode*** *(1)* selects all objects in the current layer. ***The Object mode*** *(2)* selects objects as a unit, and all control points in an object are being selected. The ***Mesh mode*** *(3)* selects ***all control points connected*** to the ***control point knot*** you click on. The ***Vertex mode*** *(4)* selects ***individual control points***, one at a time. The Vertex mode is primarily used for decomposing control point knots when you need access to individual controls.

The ***Object mode*** *(2)* has two optional methods in the ***Mode bar***. The ***left button*** *(1)* selects ***all objects*** associated with the ***control point knot*** clicked on. The ***right button*** *(2)* only selects ***one*** of the possible objects. Two ***additional arrow buttons*** *(3-4)* occur when you select this option. They ***toggle*** between all ***possible candidates***, one by one. Once the right one is found, click on the Intersection point to start dragging.

The Selection mode also affects the way new items are being added using the Option or Option/Shift keys.

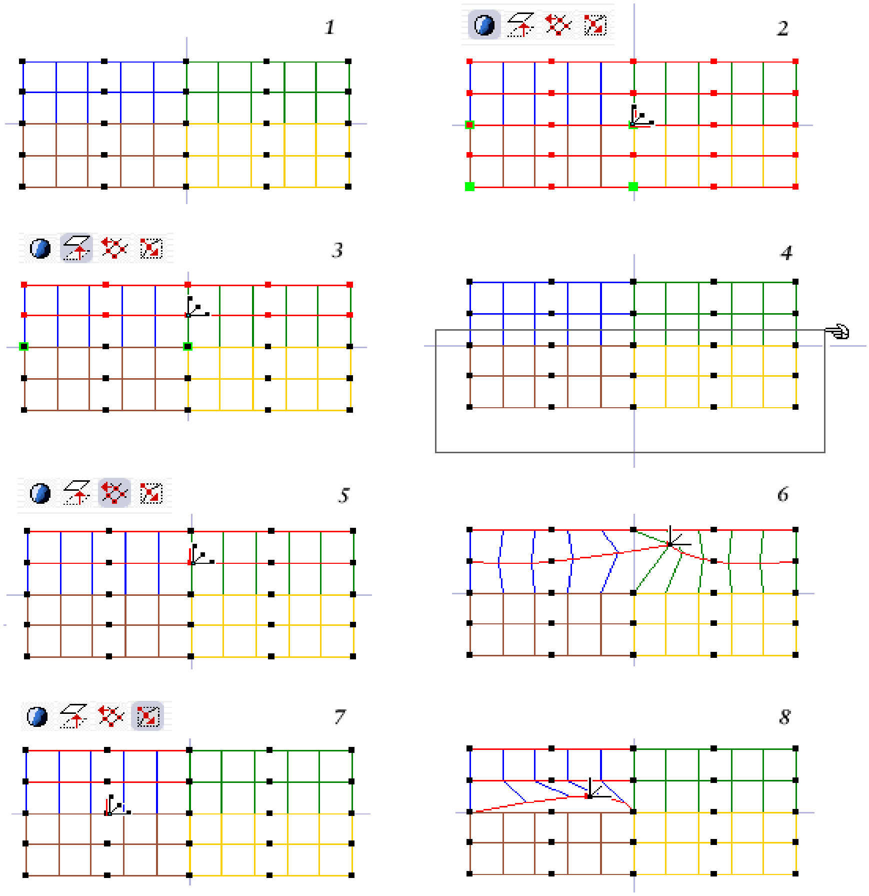
Object Mode duplicates the selected ***objects*** when you ***Alt–click*** on an object and then start dragging.

Mesh / Vertex mode adds new ***control points*** when you ***Alt–click*** on a control point.

Mesh / Vertex mode adds a new ***control ruler*** when you ***Shift + Alt–click*** on a control point and start dragging.

Selection Mode examples

The *Selection mode* options are illustrated in the picture below:



Four surfaces were drawn, each having different line colors (1). ***Global mode*** (2) selects everything in the currently active layer. ***Object mode*** (3) selects objects as units. Clicking on a control point ***along the edge (3)*** between two or more surfaces selects all surfaces connected to that ***control point knot***, in this case the blue and green surfaces. Clicking at the center-point of these four surfaces would select all four surfaces.

A ***selection rectangle (4)*** select all surfaces ***enclosed*** by the rectangle, in this case the red and yellow (bottom two) surfaces are being selected.

Group/Object mode generates a copy of the selected surface if you press the ***Alt / Opt key*** before you start to drag.

Mesh mode (5) selects all points located at the same physical location. All ***connected*** control points ***move (6)*** when dragged in this mode. ***Mesh mode*** can be used for adding new control points and control rulers.

Vertex mode (7-8) activates and moves ***one*** control point at a time.

Adding new control points on a ruler

New control points can easily be added using the following steps:

Make sure that the Selection Arrow is selected. Choose the ***Mesh*** or ***Vertex mode*** in the ***Selection mode*** bar (the two buttons to the right). The ***Object*** and ***Global*** mode will generate ***new objects*** and that is not what we want to do now. Also, you can't use grouped objects because that would not work either. Select at least one control point.

Press the ***Alt / Opt key*** and then click again on one of the selected (red) control points. ***New points occur*** when you drag them a bit. Note that the new points are added ***upwards and away*** from the ***green starting point***.

An alternative method is to use the ***Add Control Point*** command found in the ***Tool menu***. It essentially does the same thing as with the keyboard.

Is it possible to add points downward in the index list as well? Yes. Just use the ***Flip Direction -> Along Ruler*** command in the ***Edit*** menu to reverse the index order of the ruler and consequently move the green starting point to the other end of the first ruler.

Adding new rulers

The procedure for adding a new control ruler is similar to adding a point. Select a point in a ruler by clicking on it, press the ***Shift + Alt / Opt keys***, and then click on the ***control point*** again. Drag the ruler to its new location. As expected, the new ruler is being added upwards and away from the green starting point dot.

An alternative method is to use the ***Add Control Ruler*** command found in the ***Tool menu***. It essentially does the same thing as with the keyboard.

Deleting control points and rulers

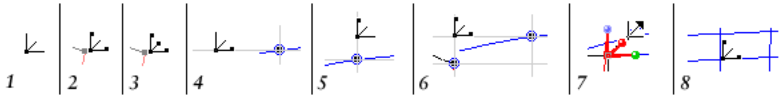
Deleting control points and rulers is done in a similar way. Select the point(s) you want to delete, and then press the ***Delete*** or ***Backspace keys*** to delete the points. A ruler is automatically deleted if all control points in the ruler are deleted.

Making a copy of an object

A copy of an object can be made is several ways.

One way is to use the ***Duplicate*** or ***Duplicate Array*** commands in the ***Edit*** menu but we will revert to that later in the manual (in the Menus section).

A quick and easy way of creating a copy of an object is to select the ***Arrow*** tool in the ***Tool*** palette, ***Object Selection*** mode, and then pressing the ***Alt/Opt*** key while clicking on the selected object and before you start dragging.



Cursors and snapping

The cursors play an important role when editing objects in TouchCAD.

The cursor shown when placing it over the drawing area symbolizes the three axes X, Y and Z. The horizontal and vertical lines represent the screen plane and the diagonal line the direction away/towards you. Small black dots occur at the end of the respective axis whenever a snap in that direction occurs. The options are:

1/ No snap
2/ 2D snap
3/ 3D snap
4/ Horizontal alignment snap
5/ Vertical alignment snap
6/ Horizontal and vertical alignment snap
7/ Move the Intersection Point to the closest control point in the direction away/towards you (note that other selected control points also move the same distance). The intermediate red dot in the diagonal line indicates where the hotspot is located.
8/ Snap to mesh grid. The cursor indicates if the snap is in 2D or 3D.

If you hold the cursor over a control point of an object, it changes and gets black dots at the end of at least two of the lines *(2-3)*. This implies that a ***snap event*** has occurred. Dots at the end of the horizontal and vertical lines mean that the snapping is on the control point in the horizontal and vertical directions.

In some cases you may see a dot on just one cursor direction *(4-5)*. This indicates that an ***alignment event*** has occurred with an alignment point (indicated by a small blue circle). This only happens if the ***Alignment Indicator*** option is activated in the ***Constraint palette*** (the bottom left button). In some cases you can get a double alignment indication. This indicates that an alignment has two directions. You move alignment points by holding the cursor over a control point for a short period of time.

If a dot occurs on the ***third diagonal line*** *(3)* you have a snap in the direction ***away / towards*** you, that is, the control point is located at the current working plane defined by the Intersection Point location, disclosed by the Intersection Point edit fields above the drawing window. The dot here means that the cursor has a ***3D-awareness*** and indicates a snapping in the direction away / towards you, which you really can't see on the screen.

If you click on a control point, the Intersection Point / working plane moves the depth location in the direction away / towards you.

If you drag a point to the location over another, a snap event occurs. The moved point gets the same horizontal and vertical coordinate location as the point it snapped to, provided that the ***2D-snap*** option is selected in the ***Constraint*** palette (the top Left button).

The snapping may be ***three-dimensional*** too if the ***3D-snap*** option is selected in the ***Constraints*** palette (top right button). This moves the dragged control point to the location of the object it snaps to in the direction away / towards you. What happens if there is more than one location along the axis away /towards you? TouchCAD simply chooses the ***closest*** point measured from the dragged point's original location in the direction away / towards you. This applies to the standard Front, Top, and Side views. In the Perspective view you always get a 3D snap because it is assumed that you then rotate the view to get a clearer view before you start dragging.

There is also an optional method that can be used for moving a selected point to the closest control point in the direction away / towards you, and ***without the 3D snap*** feature activated. Drag the cursor ***along the diagonal line*** of the ***Intersection*** point to the ***red intermediate dot***. A new arrow cursor occurs *(7)*. ***Click*** and the ***Intersection point moves*** to the ***closest*** available ***point***. Nothing happens if it can't find such a point.

The ***Resize cursor*** that looks like a double-sided diagonal arrow occurs when you hold the cursor over a ***Resize box*** control handle. A ***curved*** version occur of ***the Resize cursor*** when holding over the ***gray selection box lines*** in between the controls to indicate that a ***rotation*** will take place if you click. The curved version comes in four shapes, used for indicating the rotation direction when pressing the Alt / Opt key for rotation around the diagonal corner (as opposed to the center-point normally used).

The ***Snap to Mesh*** option in the ***Constraints*** palette (the bottom right button) allows you to drag and snap to ***individual mesh points*** in an object that are ***not control points***. The cursor is however the same as for an ordinary snap.

TouchCAD also use a few ***other cursors*** such as the ***Zoom*** magnifying glass, the ***Pan*** hand and the ***Selection Finger*** hand used when drawing selection boxes.

These features are controlled by the ***Constraints palette*** normally located below the ***Tool palette*** to the left of the drawing window. It can be made visible if invisible by selecting ***Constraints*** in the ***Palettes menu***.

Rulers, points, curves, and surfaces and definitions thereof

TouchCAD objects can be subdivided into five different sub-data types named ***Points***, ***Rulers***, ***Curves***, ***Surfaces and Groups***.

A ***point*** is an object consisting of a single set coordinates, and can described as a ruler point.

A ***ruler*** is a number of connected points all located in the ***same ruler index***.

A ***curve*** is a number of connected points, in which each point is located in ***its own control ruler***.

A ***surface*** is an object consisting of at least two control rulers, where at least one of the rulers have at least two control points (in short a triangle).

A ***Group*** is a one or several objects grouped into a unit. All control points in a group are activated when you click on a single point.

Identifying object types

It is important to understand the difference between these data types for a number of reasons. You can see the difference between rulers, curves and surfaces by selecting them.

A ***ruler*** changes color to red if you select it because it consists of a single control ruler.

A ***curve*** keeps its original color because it essentially consists of ruler starting points. Rulers in curves do not become red when you select them because each ruler consist of a single point, and a point does not have a length that can become red.

A ***surface*** changes the ruler elements to red when they are selected, whereas the mesh parts keep their original colors.

A practical use is if you, for example, draw a circle ***defined as a ruler***, you have to press the ***Alt / Opt + Shift*** keys to extend it into a tube because you then need ***to add a new ruler***.

If you, on the other hand, start with a circle ***defined as a curve*** you already have all the rulers you need. In this case you need an extra set of control points along each of the rulers, and to do that you only need to press the ***Alt / Opt key*** before clicking ***to add the new points along each ruler***.

Points are essentially ***ruler/curve starting points*** and can therefore easily be extended into rulers and curves as described above.

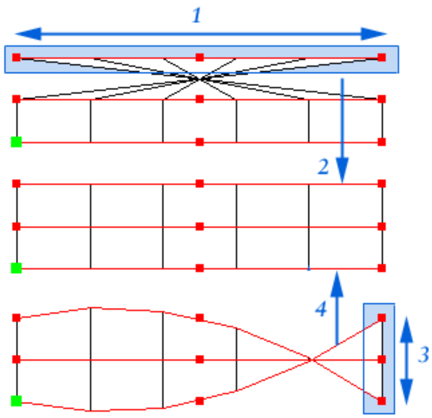
A ***group*** is simply one or several objects compiled into a unit. When selected, all control points in the group are selected. In order to edit a group you must ungroup it first using the ***Ungroup*** command in the ***Edit*** menu or activating the ***Ignore Groups*** command, also in the Edit menu.

An easy way of identifying the ***data type*** of an object is to select it, and then look at the top of the ***Object Info palette***, where the ***Object ID, Ruler ID, Vertex ID and data type*** of the object having the Intersection point is being specified.

Changing direction of an object

In some cases you may realize that it would have been better to use a ruler instead of a curve or visa versa. It can be easily be fixed by using the ***Flip Direction -> 90 Degrees*** command in the ***Edit menu***. It ***converts rulers to curves and curves to rulers***.

The ***Flip Direction -> Along Rulers*** and ***-> Perpendicular to Rulers*** commands in the ***Edit menu reverses*** the ***object direction along the rulers and perpendicular to the rulers***. A common use for such commands is when adding new points or rulers and when the object direction is wrong, that is when you want to add points downwards towards the green starting point and not upwards as you normally do. The solution is simply to change the direction of the object.



It is also possible to change a ***single ruler*** or ***curve*** within a given object, for example to fix a lofted surface where one of the rulers had an incorrect direction. The ***Flip Direction -> Selected Ruler / Curve*** commands fixes such problems.

In the ***upper example*** *(1)*, the ***upper control ruler is flipped*** in direction compared to the other two rulers causing the deformed shape. Place the Intersection Point on the upper ruler and select ***Flip Direction -> Selected Ruler*** *(2)*.to fix it

In the ***lower example*** *(3),* the ***right curve***, perpendicular to the control rulers, is flipped in direction compared to the other two curves. Place the Intersection Point on one of the control points to the right, belonging to the right curve, and select ***Flip Direction -> Selected Curve*** *(4)* to fix it.

Constraints palette

Constraints palette

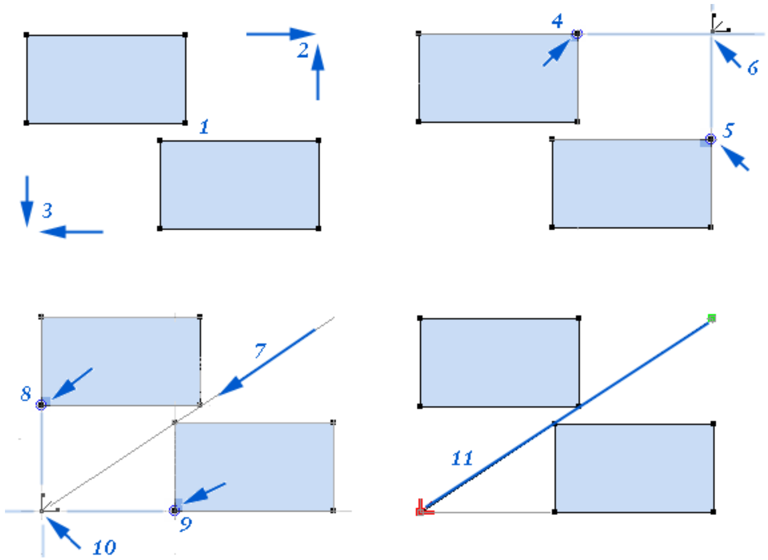
The ***Constraints palette*** controls the how objects ***snap*** when you drag them. The options are ***Snap-2D** (1)*, ***Snap-3D** (2)*, ***Alignment indicator** (3)*, ***Snap to Mesh** (4)*, ***Slide Along Ruler** (5)* and ***Slide Along Surface** (6)*.

Snap 2D

The ***Snap 2D*** mode is used for ***2D snapping*** to control points and mesh points. In the fixed ***Front, Top and Side views*** it snaps to objects ***along the current working plane*** but ***not*** in the direction ***away / towards you***. If you are in a ***Perspective*** view you will get a ***full 3D snap*** to control points because you are expected to rotate the view to a location where you can easily identify the point to snap to.

Snap 3D

The ***Snap 3D*** allows you to snap to objects in ***all three directions***, ***including*** the direction ***away / towards you***. If there are several possible candidates located at different levels in the direction away / towards you, it will ***snap to*** the point ***closest to the current working plane***. The fixed ***coordinate fields*** above the drawing area define the location of this ***working plane***. In the Front view, the working plane location is being defined by the Z coordinate, the Top view by the Y coordinate, and the Side (Right) by the X coordinate.



Alignment indicator

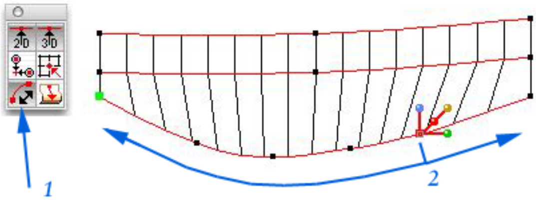
The *Alignment indicator* is an aid allowing you to place ***temporary help points*** and get ***aligning lines*** from them. Above is an example showing how to draw a diagonal line *(2-3 & 11)* based on the outer edges of two rectangles *(1)*.

Make sure that the ***Alignment indicator*** and the ***2D snap options*** are selected in the ***Constraints*** palette. Choose the ***Line*** tool in the Tool palette. Hold the cursor over the ***first point*** *(4)* used as an alignment point. ***Don't click***. A small ***circle*** occurs around that point. Hold the cursor over ***the second*** *(5)* ***alignment point***. ***Don't click***. Move the cursor to a point that roughly ***aligns horizontally*** and ***vertically*** *(5)* with the previous points. The ***cursor*** now ***indicates a snap*** in both the horizontal and vertical directions. Note that you can get ***a dot on either the horizontal or vertical cursor line ends*** and that suggest that you have an ***alignment in one direction*** only. *Click* and ***drag*** *(7)* the cursor towards the intended end of the line. Hold the cursor over the ***first*** *(8)* and then ***second*** *(9)* alignment point ***without clicking***, and then move the cursor to a point close to the ***final point*** *(10)* and then ***click***. The line should look like in the final (11) picture.

Note that the alignment Indicator can be used for aligning in a ***single direction*** and that can also be quite useful. The ***quick key is*** *(L)*.

Snap to Mesh

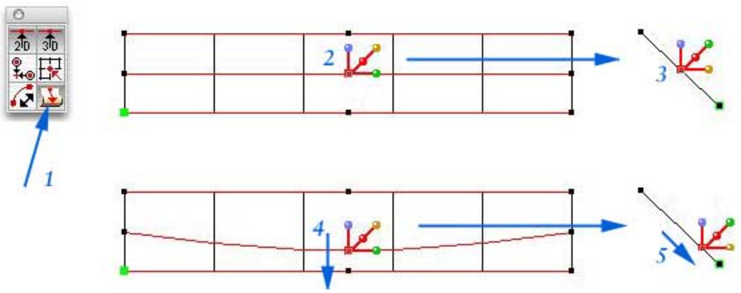
The ***Snap to Mesh*** feature allows you to ***snap*** not only on control points but also on ***individual mesh points***. This applies to ***surfaces, rulers and curves*** (ruler starting points).



Slide Along Ruler

The *Slide Along Ruler (1)* command enables you to ***slide a control point*** *(2)* along the control ruler it belongs to. The sliding follows the original shape in three dimensions as you drag along the original ruler and while getting a snap indication.

The purpose of this feature is to re-distribute the control points along the ruler. As always with mathematically generated curves, it is best to distribute the controls as evenly as possible along the Ruler to avoid tension problems and the Slide Along Ruler feature is very useful in this respect as it generates a minimum of distortion.

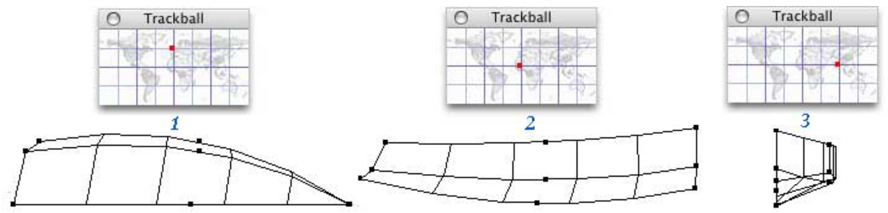


Slide Along Surface

The *Slide Along Surface (1)* feature enables you to slide a control point along the surface instead of along the fixed working planes normally used in the Front, Top and Side views. In the diagram above a control point is selected in the Front view *(2)*. When seen from the side *(3)* the surface forms a leaning plane. When the control point is ***dragged downward*** *(4)* the control point not only ***moves downwards*** but also in the direction ***away-towards*** you as seen to the right *(5)*.

The purpose of this feature is to move the control points with a minimum damage to the shape of the object. A practical use is to re-distribute the unfolded panels over the surface as these are to some extent controlled by the shape of the rulers. The practical result is a re-distribution of the unfolded patterns whereas the shape stays more or less unchanged, at least when the feature is used with moderation.

Trackball palette



Trackball palette

The ***Trackball palette*** is used for ***rotating the view***. You simply place the cursor over the palette, click and drag. A red dot occurs when you release the mouse button. The red dot indicates where on the map you are. Note that you have a faint blue world map in the background. If you are ***above the horizontal centerline*** you are on the Northern Hemisphere, that is, you ***look*** on the model ***from above (1)***. If the dot is close to the ***horizontal centerline*** you are close to the ***Front view (2)***. If it moves one quadrant to the ***right*** your are close to the ***Right view (3)***, and so on. The Trackball can be used at any time, regardless of which tool you currently use. This also applies when the model is displayed as a rendered picture.

A similar feature, called ***Rotate View***, is found in the ***Tool*** palette. The difference is that you need to select the tool first and then rotate by dragging over the drawing surface.

The ***Trackball*** palette also provides ***alternative*** features. Press the ***Shift key*** while dragging and the palette goes into the ***Dynamic Pan mode***. Press the ***Opt /Alt key*** and the palette goes into the ***Dynamic Zoom mode***.

The ***Trackball*** palette is by default set to using the ***automatic Roll-Up*** feature that rolls up the palette into strip when not used. Hold over the palette and it rolls down again automatically. The advantage is that it leaves valuable drawing area uncovered. In some cases you may want to leave it rolled down, and it can be done using the ***Trackball Roll-down*** command found in the ***Palette menu***.

The ***sensitivity*** of the Trackball palette can be adjusted by Control-clicking on it. A pop-up slider occurs below the map where you adjust the settings. Click on the Trackball again or move the cursor out of the Trackball area to go back to the normal mode.

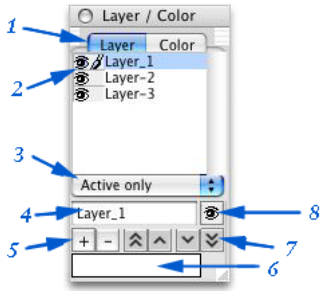
It is also possible to ***nudge rotate*** the view by means of the ***arrow*** keys in combination with the ***Opt/Alt key***.

Layer / Color palette

The Layer/Color palette

The ***Layer/Color palette*** is used for adding and deleting layers and colors and for controlling the accessibility of layers.

The ***Layer/Color*** palette is by default set to using the ***automatic Roll-Up*** feature that rolls up the palette into strip when not used. Hold over the palette and it rolls down again automatically. The advantage is that it leaves valuable drawing area uncovered. In some cases you may want to leave it rolled down, and it can be done using the ***Layer/Color Roll-down*** command found in the ***Palette menu***.



Layers

Layers can be ***visible or invisible***. Selecting a layer and then clicking on the ***Visibility button*** (*8*) changes the visibility. Alt / Opt – ***clicking*** on the ***Visibility button*** makes all layers visible or invisible. Note that the active layer is always visible regardless of the state of visibility.

The four layer-modes (*3*) are:

Active Only where you only see the active layer, regardless of the layers are visible or not.

Show All displays all visible layers but allows no snapping to objects in other layers.

Show / Snap Others displays all visible layers and allows snapping to objects in other layers but no editing.

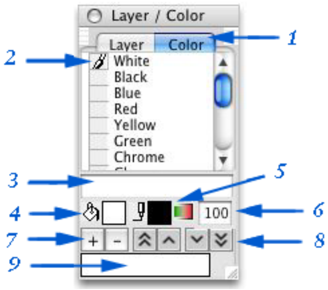
Showing / Snap / Modify displays all visible layers and allows both snapping and editing of visible layers. Changing mode can be done with a ***Visibility popup (3)*** in the Layer palette or in the ***View menu*** -> ***Layer Options*** ->.....

Layers can be ***added and deleted*** using the ***plus*** and ***minus*** (*5*) buttons in ***the Layer palette***. The ***Up and Down*** *(7)* buttons are used for changing the display order of the layers in the ***list box*** *(2)*.

Layers names can be ***changed*** it the ***name-editing field*** *(4)*. Click on the layer to be changed, write a new name and press ***Enter***.

Layers are activated by clicking on the name of the layer in the name list field of the ***Layer palette*** (*2*), or by selecting it in the ***Layer pop-up*** in the main ***drawing window***.

The ***Info field*** (*6*) displays instruction messages concerning the Layer palette.



Colors

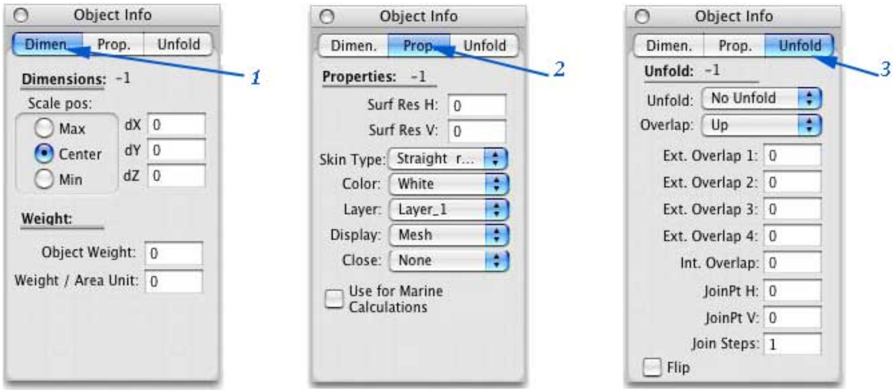
The colors used in a particular document are defined in the ***Layer/Color*** (**1**) palette. Click on the ***Color tab*** (**2**) of this palette. The picture above shows the options.

All colors have a ***name*** (**2**) to make it easier to identify them. A number of standard colors are provided when you open the program. The colors may be deleted or edited in any way you like because the color list is individual for each document. TouchCAD automatically adds any additional colors to the list if you import objects into a drawing using the ***Open Info*** command in the ***File*** menu. The currently selected color is indicated by the ***brush*** *(2)* symbol in front of one of the color. Below the color list there is a field used for ***editing the color name*** (**3**). The ***line and fill colors*** (**4, 5**) of a color definition can be edited by clicking on their respective color fields. A standard color selection dialog then occurs where you can choose any color. The line and fill colors may have different colors, which may make the model easier to understand and edit. The ***Transparency*** (**6**) field controls the ***transparency / opacity*** of the color when rendered. Note that you need an accelerated video card to be able to see the opacity when you render. Some older video cards require a color depth set to thousands (of colors) to work full-screen on a big monitor. The ***Plus and Minus*** *(7)* buttons are used for adding and deleting colors. The order of the list may be edited using *the* ***Top, Up, Down and Bottom*** *(8)* buttons. The ***Help*** *(9)* field displays short instructional messages when you hold the mouse over a field.

Below is an example showing a rendered picture containing transparent objects.

Colors can be exported as layers when exporting to DXF and VectorWorks text. This allows you to divide the model into color identities and that can be useful when exporting to a rendering program.

Object Info Palette



The Object Info Palette

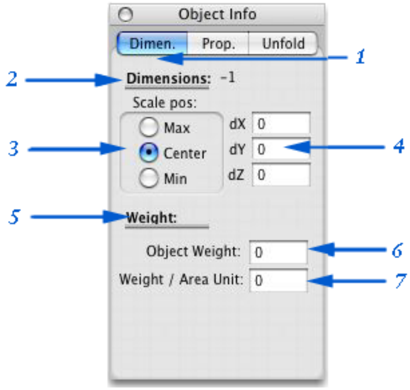
The ***Object Info palette*** is used for ***editing*** various ***object properties*** such as dimensions, weight, weight per area unit, surface display and behavior mode, and the way it should be unfolded (or not). The Object ***Info palette*** has three ***tabs***, ***Dimension*** (*1*), ***Properties*** (*2*) and ***Unfold*** (*3*).

The ***Dimension tab (1)*** controls the width, height and depth of the selected object(s). Resizing objects can be done from the lowest coordinate value, mid-point, or highest coordinate, and individually in the X, Y and Z directions. A more detailed resizing dialog can be displayed using the ***Reshape*** command in the ***Edit*** Menu. This dialog also allows scaling by a scale factor, proportional scaling, and scaling from the Intersection point.

The ***Properties tab*** (*2*) contains various object specific settings such as horizontal and vertical mesh resolution, skin type, display mode, layer and color identity, shape closing and if the object is to be used for marine calculations.

The ***Unfold tab*** (*3*) specifies the unfolding properties of the selected object(s), such as unfolding direction, if the object sub-strips should be treated as individual objects or not, if the object is to treated as a combined stretch unfold object or being joined in combined groups of objects. Overlaps can also be added individually along the outer edges of the panel, and along the internal joints. The internal joints can either be on one side, facing upwards or downwards from the green point, or be double-sided. The joining point between joined sub-panels can be set parametrically to any vertex along a joining side, and this feature can be useful when optimizing the use of material. The Flip checkbox is used for turning the selected pattern upside down when required

Object Info, Dimension Tab



The *Dimension Tab (1)* in the Object Info palette is used for defining the physical size and weight properties of the selected objects. The ***ID index, ruler index, point index*** and ***data type*** of the object holding the ***Intersection Point*** is displayed to the right of the Dimension text (2). If nothing is selected it reads –1.

The Dimensions section.

The *Dimension section (2)* allows numerical resizing of the selected objects in the X, Y, and Z directions *(4)*. The Min, Center and Max buttons controls the location of the scaling center point *(3)*. Resizing can also be done visually using the ***Resize box*** on the drawing area, and the with the ***Reshape command*** dialog in the ***Edit*** menu, which provides additional features such as resizing from the Intersection point, proportional resizing, and resizing by means of a scale factor.

The Weight section.

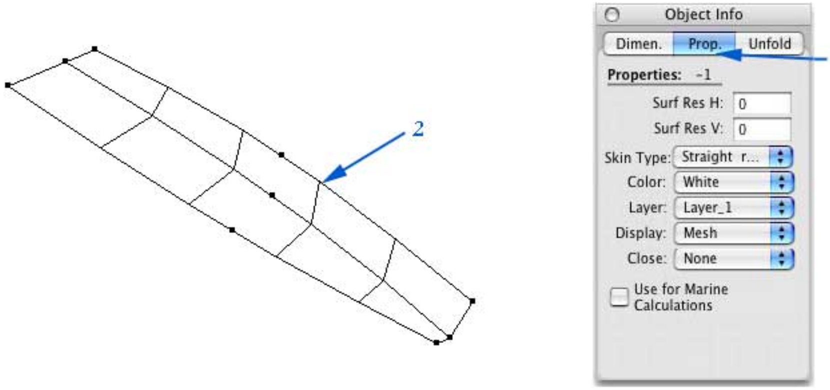
The *Weight section (5)* controls the weight properties of the selected objects. There are two methods for defining object weights.

The *first method (6) ignores the surface area* altogether. This method is primarily used when the ***true object weight*** is known, for example an engine, a battery, and other fittings. In such cases you typically know the weight of the object and in such cases you simply enter the weight of the object in the *Weight (6)* field. If a point object is used, it is assumed that the point coordinates are located at the center of gravity of the object in question. If a *surface* object is used, the *center of gravity of the surface area* is used as a *weight point*. It is a reasonable assumption provided that the object has an uniform weight distribution. If not, it may be a good idea to sub-divide the object.

The *second method (7)* defines a *weight per surface unit* property, for example 0.320 Kg / square meters or 15 Lbs. / square feet. The *Surface Properties* commands, found in the *Tool* menu, allows you to multiply this weight property with the true 3D surface area and center of gravity, to calculate the object weight as well as its center of gravity.

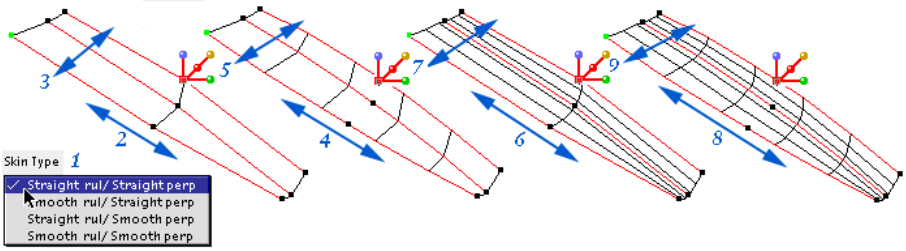
Object Info, Properties Tab

In most cases you start with a crude object sketch based on the standard tool objects, and then refine it in various ways. The ***Object Info palette*** plays a key role in this process with its many parametric settings. Let's take a closer look at what you can do with it. Start by opening the ***Manual1.T3D*** drawing. Select an Isometric view by choosing the ***Right Isometric*** view in the ***View*** menu or pressing the *(**3-key**)* on the numerical keyboard. The result should look similar to the picture below *(2)*.



The Properties tab

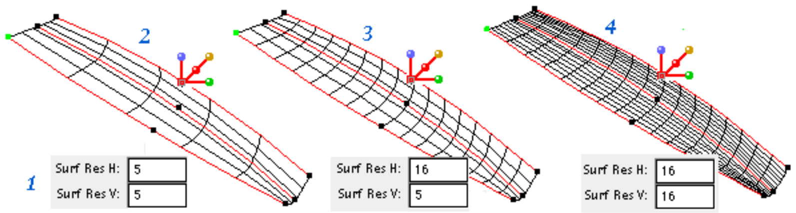
The ***Properties tab*** *(1)* is used for modifying various 3D properties in objects such as skin type, display mode, mesh resolution and so on.



Changing the surface skin type

We are now ready to experiment a bit with the object settings. Make sure that the object is selected. It is selected if it has at least one red dot along the rulers. Click on the ***Properties*** tab. Locate the ***Skin Type*** *(1)* popup.

The ***Skin type*** popup has four options. ***Straight*** *(2)* / ***Straight*** *(3)* draws straight lines between the control points in all directions. ***Smooth*** *(4)* / ***Straight*** *(5)* ***smoothes*** the object ***along the ruler*** *(4)* but draws ***straight lines perpendicular*** *(5)* to the control rulers. ***Straight*** *(6)* / ***Smooth*** *(7)* draws straight lines ***along the control rulers*** *(6)* and ***smoothes perpendicular to the rulers*** *(7)*. ***Smooth*** *(8)* / ***Smooth*** *(9)* draws ***smooth*** curves in ***all*** directions.



Changing the skin mesh resolution

Two ***Resolution*** *(1)* edit fields in the ***Properties*** tab of ***Object Info*** palette control the ***skin resolution*** setting. The default skin resolution setting is five by five steps ***(1-2).*** This may be a bit crude so try changing the ***horizontal resolution*** *(along the rulers) (3)* to 16 steps and then press ***Enter.*** Note that the object now looks smoother.

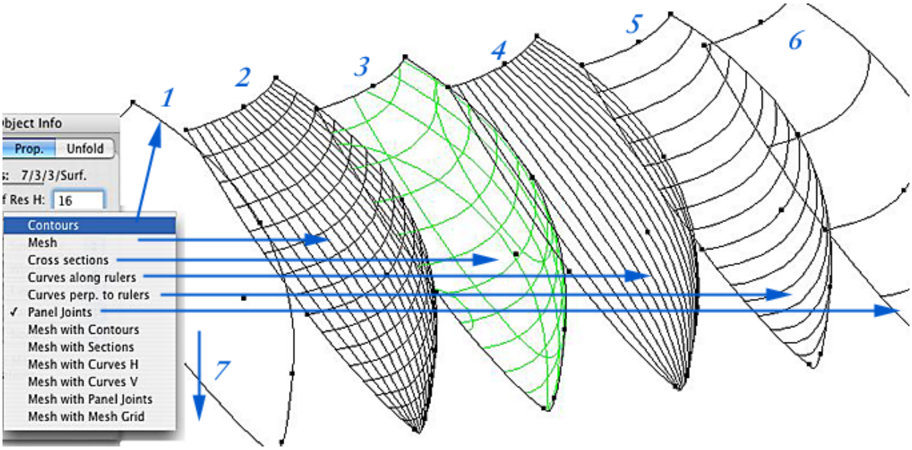
The ***Vertical*** *(4) resolution* setting (perpendicular to the rulers) is edited in the same way. Note though that ***changes in the perpendicular resolution*** are only visible in the ***skin types*** having ***smooth curves perpendicular*** to the rulers. If this happens, try changing it to Smooth along the rulers and Smooth Perpendicular to the rulers in the Skin Type pop-up. Experiment with the resolutions and skin types!

Layer and colors

The ***layer and color identity*** of ***an object*** can be changed using the respective popup menus in the Properties tab. Note that there is a difference between selecting ***the active layer and color***, which is done in the ***drawing window*** or in the ***Layer / Color*** palette and ***changing the color and identity*** of an object, which is done in the ***Object info palette***.

Display modes

TouchCAD has five display modes for surfaces. The setting can be changed using the ***Display popup*** found in ***Properties tab*** of the ***Object Info*** palette.

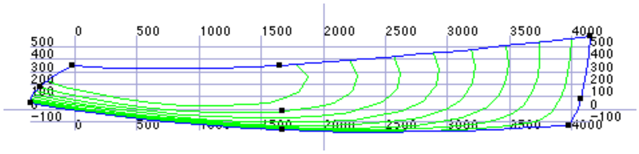


Contours.

The ***Contour mode*** (1) only displays the surface ***outlines***. The mode is primarily used for reducing the number of lines of objects that you currently are not working with but still need to see. The mode makes it easier to understand the model especially if many surfaces are used and/or if each surface has a high resolution. Each surface may consist of up to 32.000 quads or 64.000 triangles, which generates a substantial number of lines.

Mesh.

The ***Mesh mode*** (2) displays the mesh lines as well. This mode gives a good overview of how the shape looks. ***Mesh mod*** is the default mode. It displays the mesh using the specified values in the ***SurfRes H and V*** fields in the ***Object Info*** palette.



Cross-sections

The ***Cross Sections mode*** (3) displays the cross sections as shown in the picture above. The ***Cross sections*** option displays the model using ***fixed section steps***, similar to the way contour lines appear on maps. The ***cross sections*** are only ***visible*** in the fixed ***Front, Top and Side*** views. Other views are displayed as contours.

TouchCAD allows you to switch between the Front, Top and Side views and dynamically update the cross sections as you modify the shape. This provides a detailed control of the shape of the surface. It is primarily used for modeling shapes in a very accurate way, for example, to define a boat hull. It is usually recommended to limit the use of this mode, especially at high surface and sectioning resolutions because it requires quite a lot of processing power. It can therefore be a recommended procedure to use it while defining the shape of a give object and then turn if off when you're done with it.

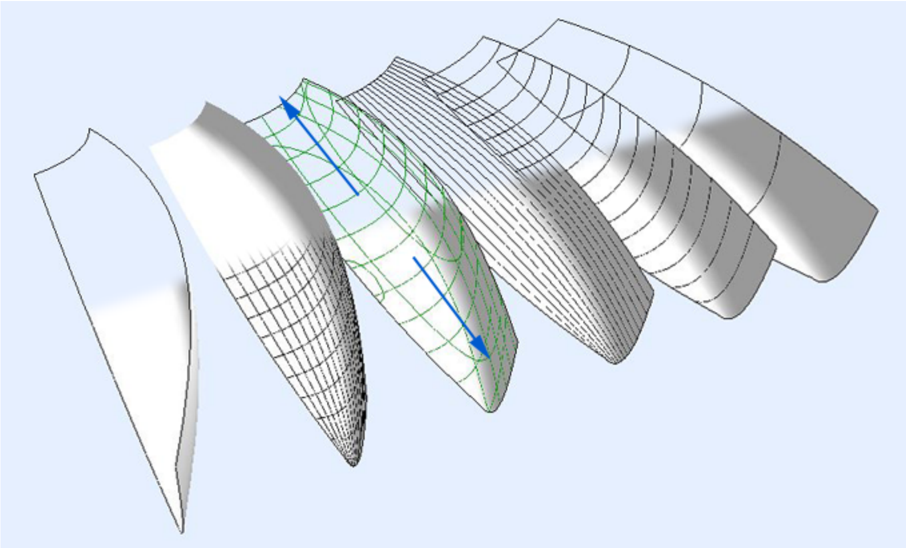
Choosing ***Sectioning Settings*** in the ***Page*** menu sets the sectioning options, including the X, Y and Z direction steps, help lines and corresponding lead texts, etc. The sectioning starts where the light blue lines are located on the screen and away there from in any direction until the model ends. The relationship between grid and model can either be changed by simply moving the model, or by moving the zero point using the ***Move Print Area*** command in the ***Page*** menu. Note that the Move Print Area setting is unique for each view.

Curves along or perpendicular to the rulers.

These two display modes are similar to the Mesh mode but the grid lines are only seen in one direction. The ***Curves Along*** (4) and ***Perpendicular to the ruler*** (5) modes can be useful for displaying the panel layout in the 3D-model while comparing it to the unfolded patterns, especially if you turn on the ***Show 3D Object ID*** feature in the ***View*** menu. It can be described as an ***assembly map*** for the ***unfolded parts***.

Panel joints.

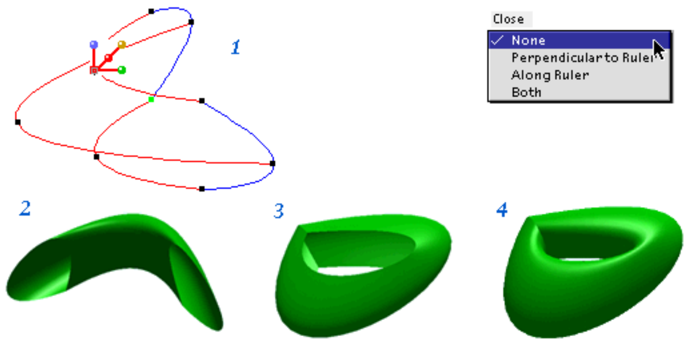
The Panel Joints mode displays where the unfolded panels joints are being located. If an object lacks unfolding properties or is set to one of the Stretch Unfold modes the panel is displayed as an outline.



Overlapping mesh.

The display modes from item 7 and below appears to don nothing in the drawing window but it affects the way objects are being displayed when rendered. All items below item 7 except Mesh mode renders surfaces as wireframe objects. Mesh mode renders surfaces as a solid skin without edge highlights as seen above in the upper section of the image. All items from 7 and above combine a skin rendering with highlighted outlines, meshes, cross sections, lines along or perpendicular to the surface direction and panel joints. The later often adds clarity to the illustrations and gives a slightly more drawing style feel to the image.

The options from item 7 also exports DXF, VectorScript and 3DMF (QuickDraw3D) files as seen when rendered, that is as a skin surface with interlaced contours.



Closing shapes

Objects can be closed in a number of ways using the *Close* popup in the *Properties* tab. Control rulers, curves and surfaces can all be closed in one way or another. A simple example is if you draw a control ruler where the end point is not connected to the starting point. It is possible to connect these points by selecting the *Close along Ruler* item in the *Close* popup. This feature connects the start and end point(s) internally. The same applies to objects consisting of curves (ruler starting points) and surfaces. Surfaces can be connected along the control rulers as well as perpendicular to the control rulers or both, which can be useful in some cases.

The example above shows a *surface (1)*.

The second *(2)* example shows the same surface *being closed perpendicular to the control rulers*. This option adds a ruler above the ruler with the highest index and physically place it at the location of the first control ruler. Note that the shape is not closed along the control ruler.

The third *(3)* picture shows the same object being *closed along the control rulers*. This option connects the first and last control points along a control ruler so that a closed shape is created. Note that it is still open perpendicular to the control rulers.

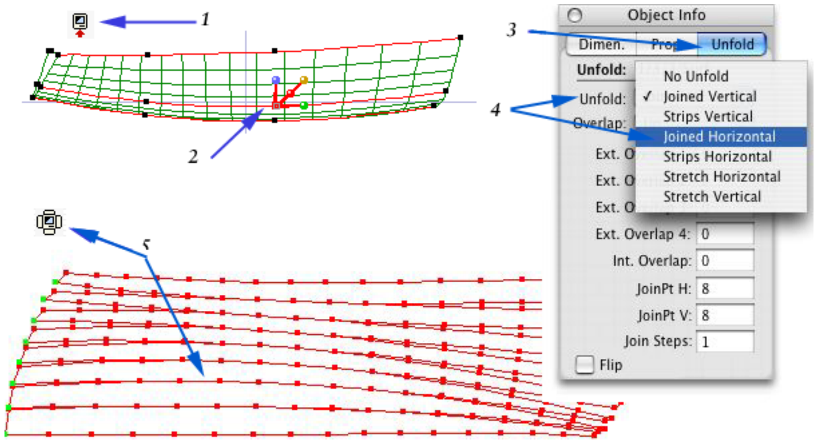
The forth *(4)* example closes the surface *along as well as perpendicular* to the control rulers. The result is *closed in all directions*.

Marine Calculations

The last item in the Properties tab defines an object as being used for marine calculations such as overall length, length in the waterline, overall beam, beam in the waterline, draft, displacement, prismatic coefficient, block coefficient, center of buoyancy, lateral center point, wetted surface, and section area curve. The calculation can be performed on any number of surfaces, but it may be practical to limit the number of surfaces to the relevant objects only. You will find more details on this subject in *Marine Properties* menu command section of this manual.

Object Info, Unfold Tab

A key feature in TouchCAD is the ability to unfold virtually any surface defined by two control rulers and at least two control points.



Unfold View

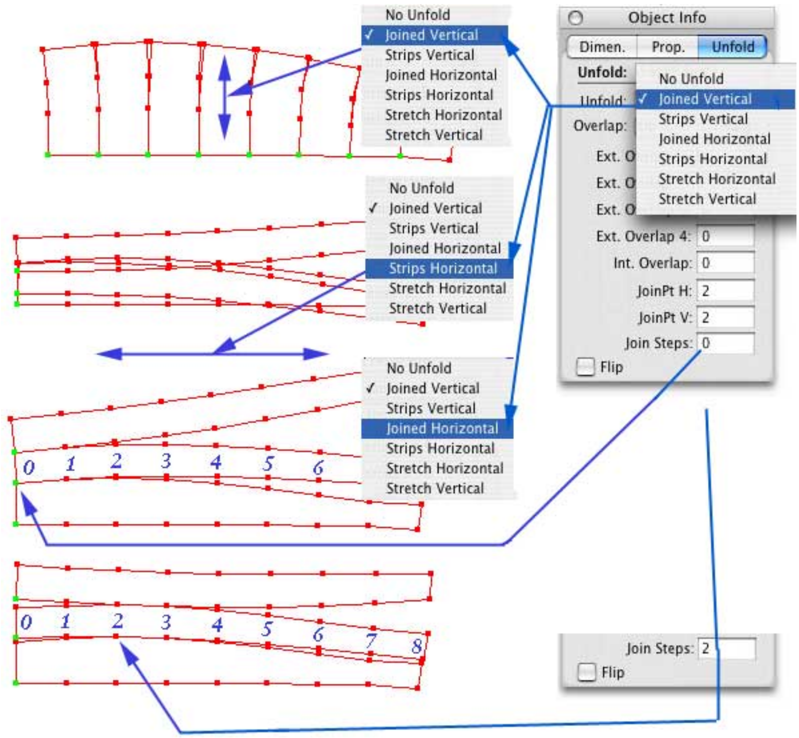
Start by selecting the object. Click on the hull surface using the ***Selection Arrow***. Click on the ***Properties*** tab (***Prop.***) and change display mode to ***Mesh*** (**2**). Set the horizontal resolution to sixteen and the vertical resolution to eight. Click on the **Unfold** tab of the ***Object Info*** palette *(3)*. The first step is to define the surface as an ***unfoldable surface***. It is done by selecting an unfold type in the ***Unfold popup*** (**4**). ***Choose Joined horizontal*** (**4**). Click on the ***Unfold*** (**5**) button in the View bar or press the ***Zero*** key on the numerical keyboard. The surface is now unfolded using joined horizontal strips, and at the resolution of the mesh. The red dots indicate that the surfaces are selected. Now try the other options in the ***Unfold*** popup. They allow you to parametrically unfold the surface in several different ways. The ***Vertical*** unfolding option may produce too many strips to be practical for production, but it is easy to fix. Just click on the ***Prop. tab*** and then change the horizontal resolution to, for example, eight, and the problem is reduced.

Stretch Horizontal and ***Stretch Vertical*** enables you to join all strips into a single unit by adding a certain degree of deformation or stretching. This can be useful when unfolding stretchable materials such as metals and some types of fabrics, where you start with a basic sheet that you deform into its final shape. When using stretching you add tensions to the material and it should therefore be used with caution and based on experience with the materials used. TouchCAD does however provide tools to analyze and help reducing the tensions when shaping an object. More on this in the General Unfolding Settings section.

TouchCAD also provides methods being a cross between the ordinary unfolding and the stretch Unfold methods by joining sub panels in sub-groups.

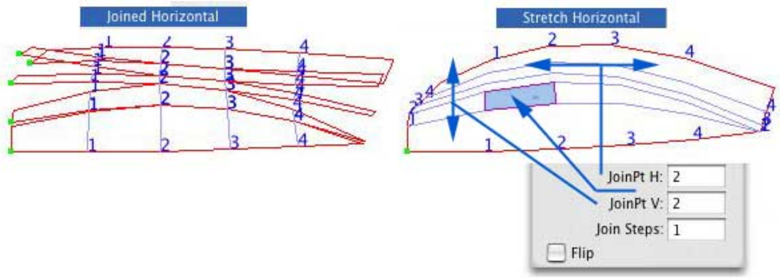
Unfold direction

Below is a picture showing what the ***Unfold*** options look like. The ***Join at Vertex*** option allows you to move the joining point between sub-surfaces by entering a factor. Zero means the first point, one the second point and so on. It is useful for making joined unfolded surfaces as compact as possible.



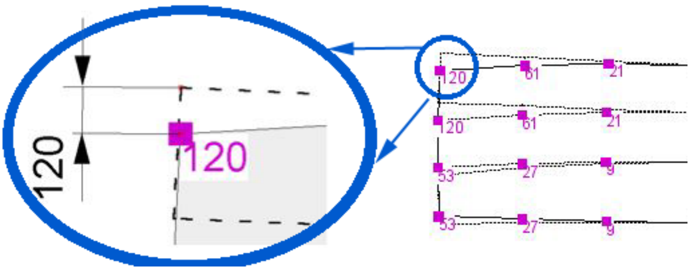
Stretch Unfold calculation

The Stretch Unfold features combine main panel sub-strips into a single unit, where you let the material carry the tensions generated by stretching the material. The result can be defined as an approximation of the final shape such as it looks before the stretching takes place. A sweater, for example, is essentially a flat piece of fabric that stretches into the final 3D shape when you take it on. The image below shows the same surface as ordinary horizontally unfolded panels (left) and as a stretch Unfold panel (right).



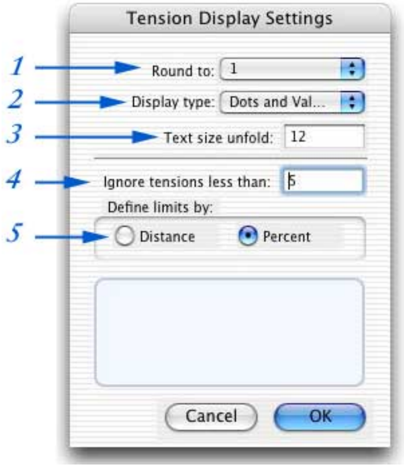
The stretch unfold options enable you to numerically place a ***balance point*** from where the calculation starts. In most cases it is best to start in the middle but when the surface has lots of tensions on one side it may be better to move the starting point to balance the tensions. The starting point is represented by a ***purple colored rectangle*** and the ***JoinPt H*** and ***JoinPt V*** edit fields in the ***Unfold tab*** of the ***Object Info*** palette is used to control the location.

Since the stretch unfold options is a form of approximation of the unfolded shape, it of course generates tensions. The image above shows the difference between the normal unfolds mode shown as dotted lines, and the Stretch Unfold mode shown as a gray surface.



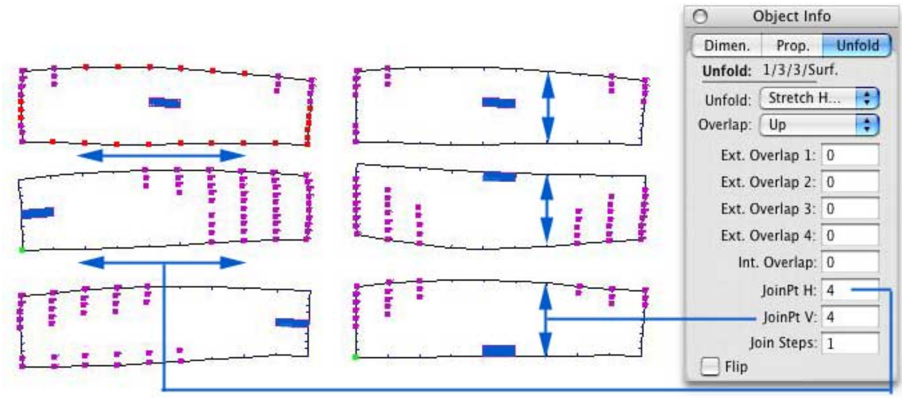
TouchCAD can however specify where the tensions occur and the degree of tensions generated. In the View menu, select ***Show in Unfold -> Stretch Tensions***. The illustration above shows the tensions defined as ***absolute offsets*** relative to their respective original location with the Stretch Unfold options turned off. The dots indicate that the vertex has an offset above the stated limit as defined in the Stretch Settings dialog found in the View menu -> Show in Unfold -> Stretch Settings.

What an acceptable level of tension is depends on the material used. A very flexible material such as the fabric used in for example a sweater of a T-shirt can be stretched quite a lot whereas for example carbon fiber or kevlar is extremely rigid. It is therefore important to use these features with caution and based on experience.



Stretch Unfold Settings

The stretch unfold settings can be found in the ***View*** menu -> ***Show in Unfold*** –> ***Stretch Settings***. The ***Round To (1)*** pop-up defines how many decimals to be used in the texts. The ***Display Type (2)*** popup sets the display style for the tension markers. The options are Dots where all points having a value above the limits are marked with a dot, Values where the same points are marked with a text label, or Dots and Values where both are being displayed. The ***Text Size Unfold (3)*** edit field defines the text sizes used within the Unfold view. The ***Ignore Tensions Less Than (4)*** edit field defines the lower limit of the tensions where they are ignored. This setting should be set according to the material used, the size of the object and so on. The ***Define Limits (5)*** buttons define if absolute values should be displayed or if it should be displayed as a percentage of the width of the local panel.



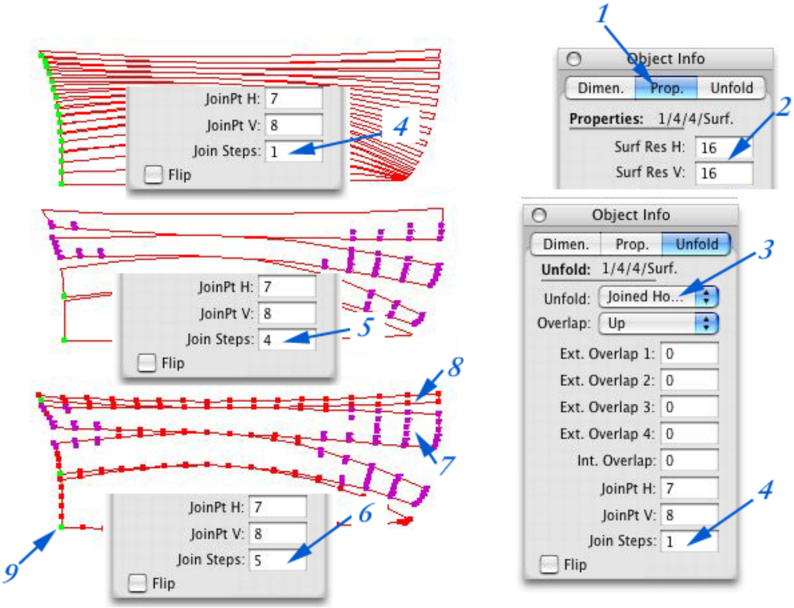
Balancing point

The balancing point is adjustable using the JoinPt H and PoinPtV edit fields in the Object Info palette as seen above, here illustrated with a filled rectangle, normally not used in the program. Note that the purple colored tension markers move as the balancing point moves. In most cases and by default it is set to the center of the surface but in some cases there is a distinct advantage to move it, for example when there are lots of tensions on one side or corner and less in other areas. It is therefore advisable to experiment a bit to generate an even distribution.

Note that the tension values change as the 3D shape changes so it may be a good idea to experiment a bit with the shape to reduce tensions as much as possible.

Joining unfolded strips into groups

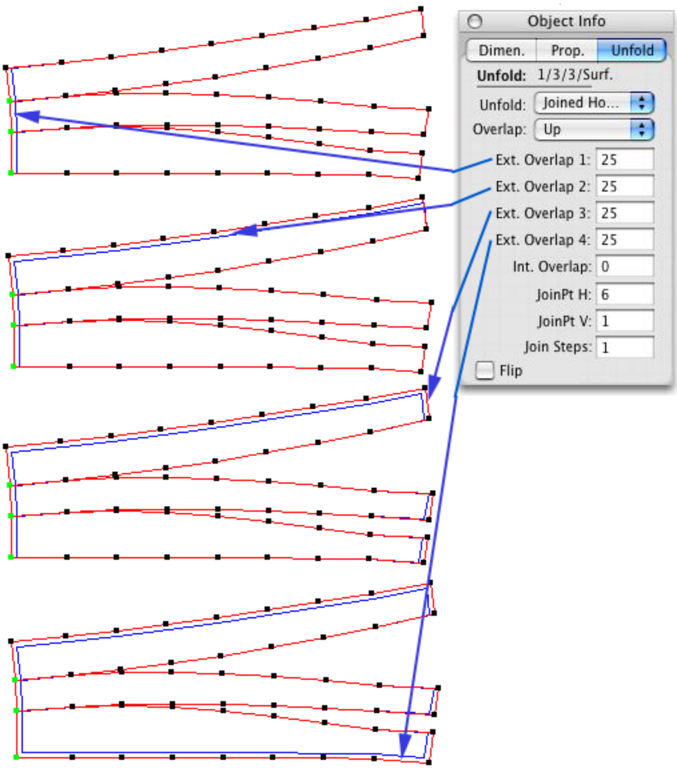
It is also possible to join unfolded strips into groups, which can be said to be a cross between the ordinary unfolding modes and the Stretch Unfold modes. This intermediate method can generate tensions just as the Stretch Unfold modes so it also supports the tension analyzing methods mentioned above.

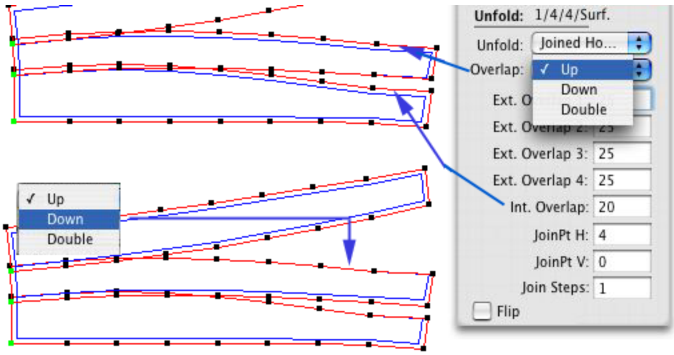


In the example above, the panel ***mesh resolution*** is set to 16 x 16 *(2)* in the ***Properties tab*** *(1)* of the Object Info palette. The ***unfolding mode*** is set to ***Joined Horizontal*** *(3)* in the ***Unfold Tab***. The edit field at the bottom of the Unfold tab controls the number of joining steps (4). When set to one (4), it splits the sub panel according to the mesh resolution (2). When set to something evenly dividable to the mesh resolution such as four (5), sixteen is evenly dividable by four, the sub-strips are joined evenly in groups of four. Note that the tension markers are now visible. When set to five (6), which is not evenly dividable by sixteen, the first three are joined in groups of five (7) and the last spare (8) forms a single strip, all counted from the green starting point (9).

Overlaps / Seam allowances

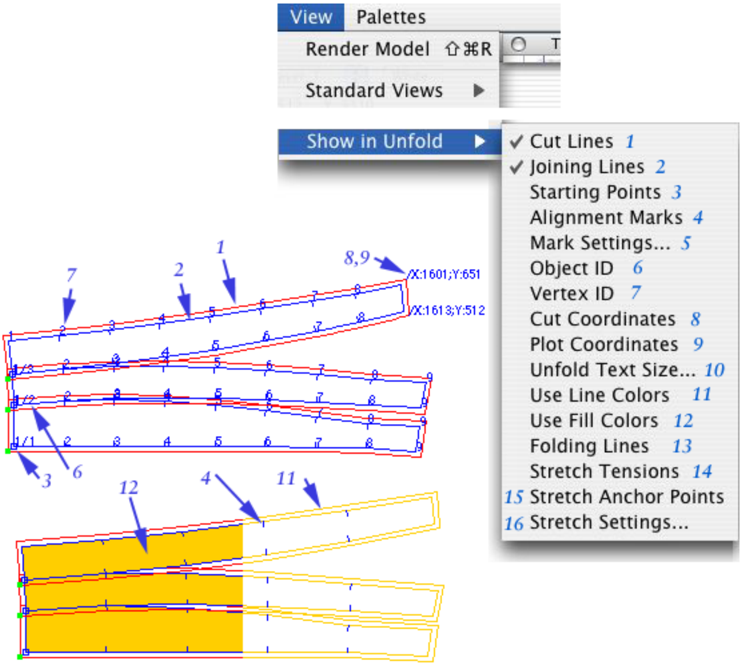
The picture below shows the *overlap* options. Each of the four main panel sides can have its own overlap setting called *External Overlap 1-4* and the joints between the main panels has a setting called *Internal Overlap*. This inner overlap can be place on either the upper or lower side seen from the green starting dot, or it can have an overlap on both sides.





General unfolding settings

TouchCAD provides a range of ***general unfolding settings***. These options can be found in the ***View menu -> Show in Unfold -> etc.***



These options allow you to organize and keep track of very large and complex models consisting of hundreds of sub-panels. All these unfold options are supported when exporting as DXF (supported by most CAD programs), VectorWorks (VectorScript) and PlotMaker (PlotMaker is a program for controlling sail panel cutting machines used by many sail makers).

1/ The ***Cut lines*** option shows the outer edge of the surface including overlaps.

2/ The ***Joining Lines***, show where the final assembled surface edges meet.

3/ The ***Starting Point*** is used for specifying where each panel starts. The point of a main panel is the green dot and it is visible in 3D too. Note that there is a Show Object ID higher up in the menu and this option allows you to see the Object ID number in 3D as well.

4/ Alignment marks (sometimes also called strike up marks) plot lines along the edge of the panel and helps to align surface edges when assembling panels.

5/ The ***Mark settings*** specify the length of the alignment marks.

6/ Object ID inserts a panel number in the corner in the following format (5/3). The ***first number*** specifies the ***main panel number*** and the ***second*** is the ***sub panel number*** (e.g. 5/3 means that the main panel number is five and it is the third sub panel in that main panel. Note that the sub-panels always occur in consecutive order with the starting point in the same corner. This keeps track of where they belong.

7/ The ***Vertex ID*** plots the vertex number along each side. This aids when assembling large panels with many points along the edge.

8/ The ***Cut Coordinates*** option plots the coordinates along the true edge of the panel including overlaps. It is useful when optimizing the use of material to fit into a given material width or height. Note that each panel has its own local coordinate system, and it will dynamically update as you move and rotate the object. It even allows you to edit the 3D-model with the arrow keys while being in the Unfold view. It works because the 3D model is dynamically linked to the unfolded model, so changes in 3D can instantly be seen in the Unfold view. If you look closely you will se that the unfolded shapes and coordinates slowly changes as you move. Optimizing the use of material can therefore be an integrated part of the design process in an efficient and powerful way.

9/ The ***Plot Coordinates*** are basically the same as the Cut coordinates but they represent the inner joining line instead.

10/ Text size sets the size of the texts in the Unfold view.

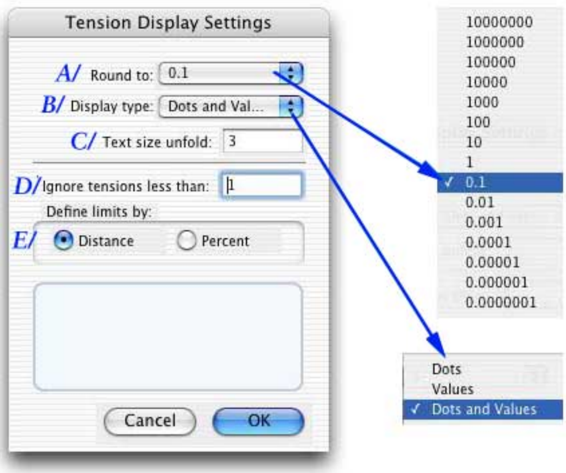
11/ Use Line Colors display the 3D-line color instead of the standard unfold colors.

12/ Use Fill Colors display the 3D fill color instead of a white outline. It can be used for printing out color scale models on a color printer.

13/ Folding lines display lines across the unfolded surfaces. The primary use is to indicate where to fold panels using a folding or plate rolling machine.

14/ Stretch tensions displays the amount and location of the tensions generated when using the Stretch Unfold modes and when combining strips into groups using the Join Steps option.

15/ Stretch Anchor Points is used as a balance point for the Stretch Unfold modes. The ***Anchor Point*** is displayed as a purple rectangle (above shown filled to make easier to find). The point should normally be placed in the middle though in some cases it may be useful to move it a bit to re-distribute and balance the tensions more evenly over the surface. This can be done using the ***JoinPt H*** and ***JoinPt V*** edit fields in the Unfold Tab of the Object Info palette.



16/ ***Stretch Unfold Settings*** is used for customizing how the tensions are being displayed. The image above shows the dialog. ***The Round To*** *(A)* pop-up controls the number of decimals used. The ***Display Type*** *(B)* pop-up controls what to draw in the drawing area. The ***Text Size Unfold*** *(C)* edit field controls the text size used in the Unfold view. It is usually practical to keep it as small as possible as the tensions calculations tend to generate lots of texts. The ***Ignore Tensions Less Than*** *(D)* edit field controls what to ignore in tensions. This setting is very individual and must be set based on experience and the type of material used. Typically, more flexible materials allow a higher setting. The ***Define Limits By*** *(D)* buttons control whether to use absolute value offsets when plotting or an offset as a percentage of the sub-strip width.

Unfold export limitations.

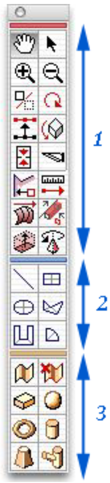
All these options can be printed and exported to other programs. There are however some limitations:

The DXF export does not support fills.

The PlotMaker and HPGL export does not support fill colors and coordinates.

The VectorWorks export supports all options.

Tool palette



The Tool palette contains various tools for drawing and editing objects. The palette is sub-divided into three sections.

The ***red section*** *(1)* at the top contains tools for editing objects and views. It includes zooming, panning, selection, rotation, cutting, joining, alignment, lofting, extruding and sweeping tools.

The ***blue section*** *(2)* in the middle contains basic 2D style tools such as lines, rectangles, circles, double lines, polygons and arcs.

The ***yellow section*** *(3)* contains various pre defined 3D objects such as surfaces, 3D points, cubes, spheres, torsos, cylinders, tubes, rectangular to circular transformations, T-tubes, etc.

Selection Arrow

The Selection Arrow tool is used for several things.

Starting and finishing activities.

First of all the Selection Arrow tool is the start and stop point of most activities. If you for example draw a rectangle, you select the Rectangle tool and draw one or several rectangles, and then finish the sequence by selecting the Selection Arrow. If you by mistake forgot to select the Selection Arrow tool and therefore started a new activity sequence with another tool, it is possible to cancel it by pressing the Escape key. If no activity are in the works the active tool shifts to the Selection Arrow when the Escape key is pressed.

Selecting and deselecting tools and activities.

The most common use of the Selection Arrow tool is to select and deselect objects, tools and activities. You typically select objects by clicking on them with the mouse.

In the drawing area you deselect by clicking on an area beside the available objects. Object can be selected by clicking on a control point or by drawing a selection box around the object or objects. Several objects can be selected by pressing the Shift key while clicking or drawing a selection box.

Editing selected objects.

Selected objects within the drawing area can also be edited in a wide range of ways with the Selection Arrow tool. The selection can be dragged, resized, scaled, deformed, skewed and rotated.

More info regarding editing with Selection Arrow can be found in the Drawing Elements chapter.

Zoom and Pan

The ***Zoom** (2-3)* and ***Pan** (1)* tools behave like a standard drawing program. There are however a few things worth noting.

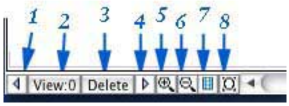
Pan tool

The ***Pan/Hand** (1)* tool in the ***Tool*** palette allows you to move the drawing area relative to the drawing window.

Panning can be done using the ***scroll bars*** along the right and bottom edges of the drawing window. A third option is to press the ***Space Bar*** key on the keyboard. This feature temporarily activates the ***Pan/Hand*** tool. Move the drawing area to a new location and press the ***Space Bar*** again to return to the previously selected tool. A forth option is to press the ***Shift key*** and then use the ***Arrow*** keys. It ***scrolls the view 1/3 of the screen width*** in the direction of the arrow.

Zoom in tool

There are no fixed zoom steps as sometimes found in other programs. Instead you have access to a powerful proportional zoom tool. The ***Zoom in** (2)* tool in the ***Tool*** palette allows you to draw a selection rectangle around the area concerned. This area will zoom in filling the drawing area. A small selection area will drastically increase the degree of zoom but choosing a selection rectangle slightly smaller than the drawing area creates a small increase in the zoom. You can ***double-click*** on the ***Zoom In** (2)* tool to double the degree of zoom.



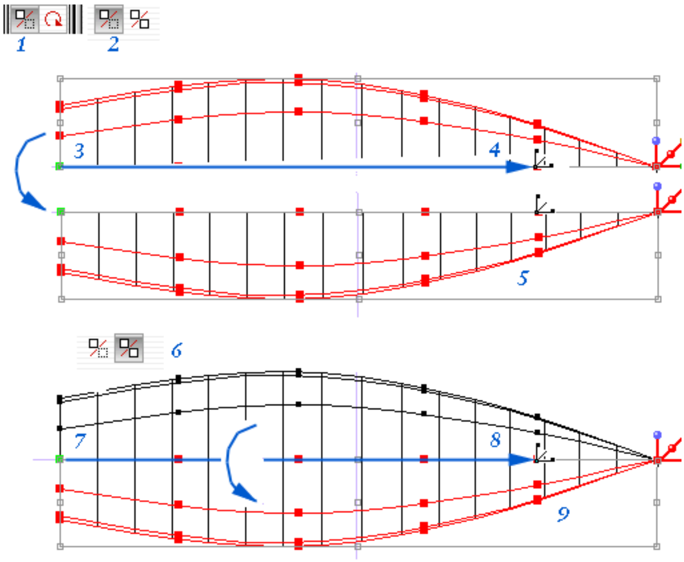
Zoom Out tool

The ***Zoom Out** (3)* tool works in reverse. The quickest way to zoom out is often to use the ***Fit to Window*** command in the ***Page*** Menu (***Command/Control + 4***), which brings back a view where the entire ***sheet*** area is seen (the light blue rectangle).

There are also some view control tools in the bottom left corner of the drawing window. The first four buttons *(1-4)* are used for storing up to ten views and to step up and down in the view list. The forth and fifth buttons *(5-6)* double and half the degree of zoom. The blue sixth button *(7)* fits the drawing area into the drawing window. The last button *(8)* centers the drawing area on the object(s) selected.

Mirror tool

The ***Mirror tool*** *(1)* is used for flipping an object around a given axis. Two modes allow you to just mirror the object or creating a mirror copy of the selection. Below is a picture showing the steps.



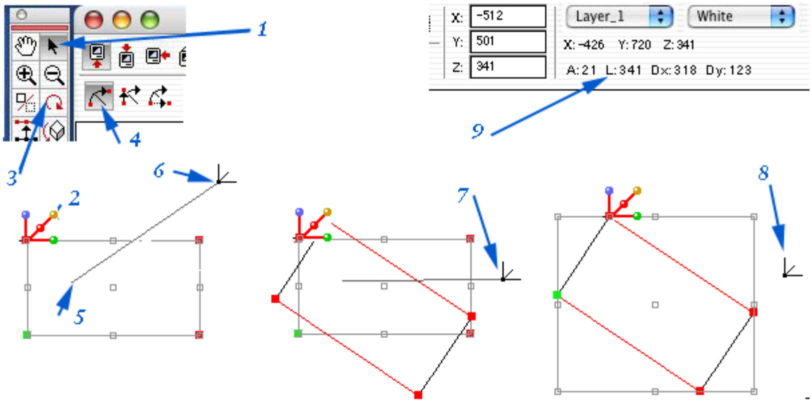
Mirror mode

In the first example we will just mirror the object. Start by selecting the object using the ***Selection Arrow***, ***Object mode*** in the ***Mode*** bar. Note that you can mirror element within an object, but that is not what we want to do now. Select the ***Mirror tool*** *(2)* in the ***Tool*** palette. Select the ***Mirror mode*** *(2)* option in the ***Mode*** bar. Draw a horizontal ***lever arm*** along the centerline by clicking on the ***first point*** *(3)* and then ***the end point*** *(4)*. In this case we used the snapping features to get a horizontal line. It is possible to draw a horizontal line by pressing the ***Shift*** key while drawing the lever arm. The mirror action takes place directly after the second click *(4)*. The picture *(5)* may incorrectly suggest that there is an offset. It has been offset for diagrammatic purposes only. The mirrored object will be located on the other side of the lever arm.

Mirror and Duplicate mode

The ***Mirror and Duplicate mode*** works in the same way. Select the object. Make sure that the ***Mirror and Duplicate mode*** *(6)* button is selected, draw the ***lever arm*** *(7, 8)*, and ***a mirror copy*** occurs on the other side of the lever arm line *(9)*.

Rotate tool



The ***Rotate tool*** *(3)* rotates selected objects. Start by selecting an object using the ***Selection arrow*** *(1-2)*. Select the ***Rotate tool*** *(3)* in the ***Tool*** palette and make sure that the ***Standard mode*** *(4)* is on in the ***Mode*** bar. The procedure consists of three steps. The first step is to ***click*** on the ***center point*** *(5)* of the rotation. A gray rubber band line occurs. Move the cursor to a ***second location*** *(6)* and ***click*** again. The line you just created is the lever arm used for the actual rotation of the object. The selected object or objects now follows the ***lever arm*** *(7)* movement around the center point. The third step is to finalize the rotation by a ***third click*** *(8)*.

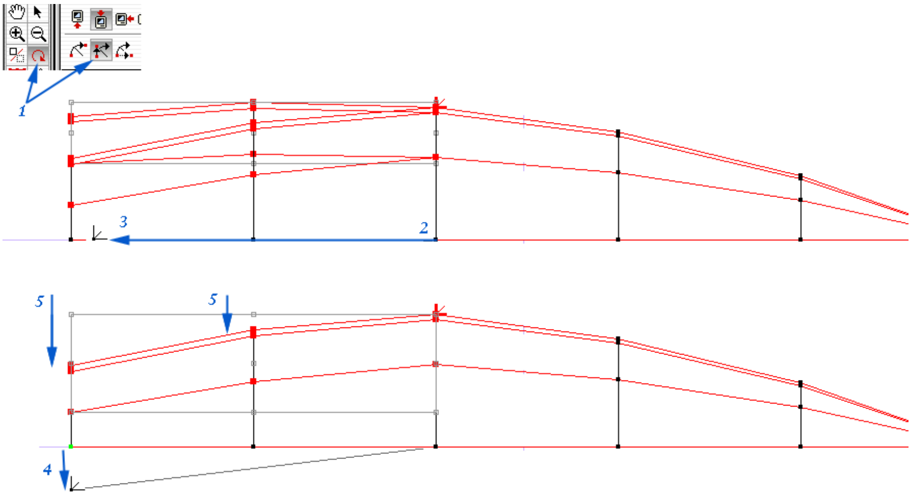
A few points are worth noting. You can see the current ***rotation angle*** *(9)* above the drawing area. You can make use of the snapping while rotating by holding the cursor over a non-selected point. You can lock the lever arm to be ***horizontal, vertical*** or being at ***45 degrees*** by pressing the ***Shift*** key. Note - all angles are in degrees reading counter clockwise from zero at due east.

Rotation modes

TouchCAD provides three ***rotation modes,*** which can be selected in the ***Mode*** bar above the drawing window. The first rotation mode rotates objects in a fairly standard way.

The ***Constraint Rotation modes block*** out the ***offsets in one direction*** and therefore result in a ***skewed rotation***. The ***Horizontal*** mode ***blocks the vertical offsets*** and vice versa.

An example of use for the constraint rotation mode can be to make a boat hull wider (seen from the top) by selecting all points from the ***mid-point*** and ***backwards*** (excluding the points along the center-line).



Select the ***Rotate tool*** *(1)* in the ***Tool*** palette and then select the ***mid-button*** in the ***Mode*** bar. This mode allows ***vertical*** movements and ***blocks horizontal movements***. Click on the ***center of rotation point*** *(2)*. Drag to the left while pressing the ***Shift*** key to lock the lever to being horizontal and then ***click again*** *(3)*. Drag the lever arm ***downward*** *(4)*. Note that the movement gets bigger further away from the center-point *(5)*. Click to finish the sequence.

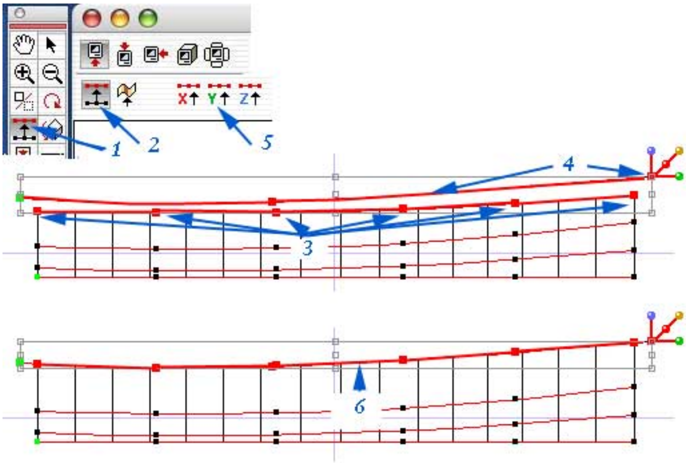
Spend time experimenting with constrain mode rotation. You will find that it can be used for generating all sorts of useful effects. An entire object can for example be skewed if you select all control point in an object and then rotate from a point close to the center of the object.

Other rotating methods

Note that TouchCAD also provides a range of other tools for rotating objects in the ***Edit menu -> Rotate -> submenus***. The options are***; Rotate Left 90°, Rotate Right 90°, Flip Horizontal, Flip Vertical***, and ***Rotate*** by means of a ***dialog***. The later provides the same features as on the drawing, but here you enter the rotation values numerically.

Align tool

The Align tool is used for aligning objects to rulers or surfaces. Two methods are provided.

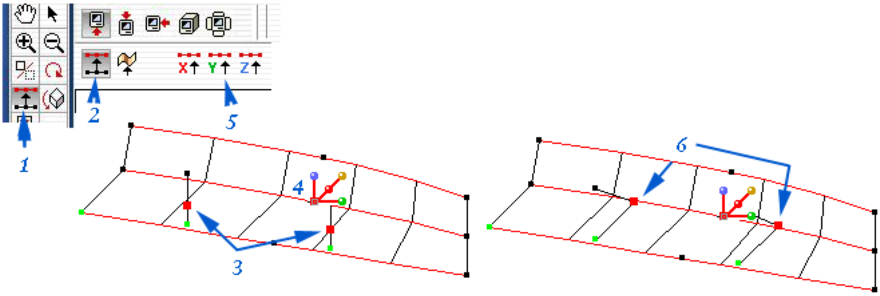


Align to ruler

The Align to ruler mode can be used for aligning control points to a control ruler in the X, Y and Z direction. Both individual rulers and rulers within a surface can be used as master curves. Below is a picture showing the steps.

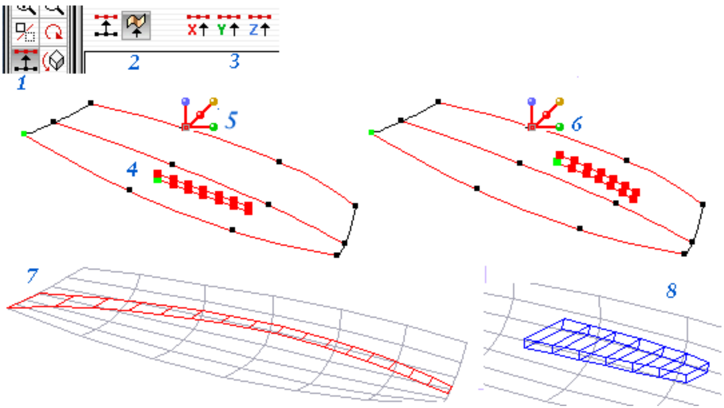
Start by drawing a ***control ruler (4)*** to which it is intended to align to. Note that is ***must be a ruler*** and not a curve (a series of ruler starting points). Select the ***control points (3)*** that you want to align to the curve, using the Selection Arrow, and then place the ***Intersection Point*** at the ***master curve (4)***. Note that you can move the Intersection point to the right curve if it is in the wrong place by ***double-clicking on a selected control point*** on the master curve if you have a selected point, or ***Shift-clicking*** on it if you have ***no selected points*** there. Select the ***Align tool (1)*** in the ***Tool*** palette, make sure that the ***Align to Curve mode (2)*** is on in the ***Mode*** bar, and then click on the related ***Directional (5)*** button in the ***Mode*** bar. The ***Directional*** (X, Y, Z) buttons ***execute*** the command ***when you click*** on it and ***align*** the ***selected points*** to the ***master curve (4)***. The result can be seen in the lower picture ***(6)***.

Individual ***control rulers in*** a ***surface*** object can also be used as a ***master-aligning*** object. The picture below shows an example where ***two control points (3)*** are being aligned to the ***center-ruler*** of the ***surface (4)***. The steps are the same as before. ***Click*** on the ***points to be aligned*** using the Selection Arrow (3) and then ***select the master ruler (4)***. Select the ***Align (1)*** tool in the Tool palette. Make sure that the ***Align to Curve (2)*** is selected in the ***Mode*** bar. Click on the appropriate ***directional button (5)*** in the ***Mode*** bar. In this case it was the ***Z button***. The selected control points align to the ***master curve (6)***.



Align to Surface

The ***Align to Surface*** mode is used for ***projecting control points to a surface***. Below is a picture showing the steps.

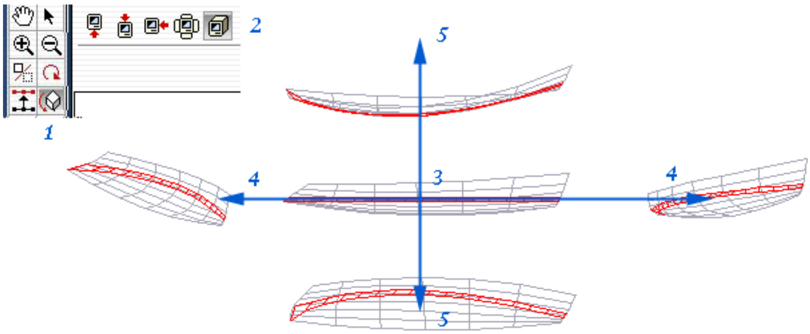


Select the ***control points (4)*** you want to project to the surface using the ***Selection Arrow*** in the ***Tool*** palette. Place the ***Intersection Point (5)*** on the ***master surface or object group consisting of surfaces (5)***. Select the ***Align tool (1)*** in the ***Tool*** palette. Select the ***Align to Surface Mode (2)*** in the ***Mode*** bar. Click on the appropriate ***Direction button (3)*** in the ***Mode*** bar to project the points on the surface. The ***result (6)*** is shown.

The ***Align to Surface*** mode can be used for many purposes such as placing graphics on a surface ***(7 - a water line)***, or to find the intersection between the surface and an internal object such as a ***bulkhead*** or a ***cushion (8)***.

Rotate View tool

The Rotate view tool rotates the view. It does not rotate the objects. You rotate the view by selecting the ***Rotate View (1)*** tool, note that the ***View mode*** changes into ***Perspective mode (2),*** hold the cursor over the ***drawing window (3), click and drag (4-5).*** Drag horizontally to rotate ***horizontally (4)*** and vertically to rotate ***vertically (5). Click*** again to finish the sequence.



Other ways to rotate the view

Another method for rotating the view includes using the ***Trackball*** palette, which has the advantage of being modeless and can therefore be used with most other tools without switching tools. Other methods include using the ***numerical keyboard (0-9),*** and using the ***standard views*** in the ***View*** menu. The view can also be ***nudge rotated*** using the arrow keys in combination with the ***Alt/Opt key***.

Join tool

The ***Join (1)*** tool is used for joining objects. Two objects can be joined provided it has the ***same number of control rulers*** and that they are ***physically connected along one edge*** and that the ***rulers have the same direction***. ***Surfaces***, as well as ***curves*** (a series of ruler starting points), and ***control rulers*** can be joined.

Below is a picture showing the steps.

Select the ***Join (1)*** tool in the ***Tool*** palette. Click on the ***first control point (2) along the joining side*** and then on a ***second control point (3)***. The objects are ***joined (4)*** provided all the above mentioned conditions are met.

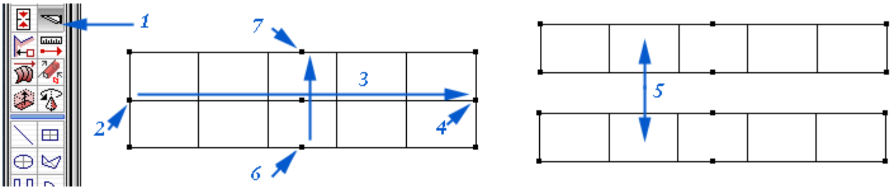
A few things are worth noting. The reason for having to click on two points is that it allows you to join two objects in the middle of a bigger mesh of individual objects. The joining lines typically reduce the number of possible joining candidates to two. When ***joining curves*** (a series of ruler starting points) and ***control rulers*** you don't have access to two individual points. In such cases it is perfectly valid ***to click twice on the same point***, because you then seldom need to worry about other objects.

Other joining methods

TouchCAD also allows two objects to be joined using the ***Join menu command*** in the ***Edit*** menu. In this case you simply ***select the two objects*** to be joined and then select the ***Join*** command to join them.

Cut tool.

The *Cut tool* is used to divide objects at ***control points***, ***along control rulers*** or ***perpendicular to control rulers***. The method used is very similar to the Join tool. Below is a picture showing the steps.



Cutting along a ruler

Select the ***Cut*** *(1)* tool in the ***Tool*** palette. Click on the ***first cutting point*** *(2)*. A ***rubber band line*** *(3)* occurs that you drag to the ***second cutting point*** *(4)* where you ***click*** again. The screen flickers slightly indicating that the cutting procedure has taken place. Use the ***Selection Arrow*** tool (***Object*** mode) in the ***Tool*** palette to ***separate*** *(5)* the two elements.

Cutting perpendicular to a ruler

Cutting perpendicular to the control rulers works in the same way. Use the ***Cut*** *(1)* tool and draw a ***cutting line*** *(6->7)* ***perpendicular*** to the control rulers.

Placing a cutting line along the edge of for example a ruler generates an error message. The reason for this is of course that it doesn't make sense to do a cut if you don't get anything on both sides of the cut.

Cutting off control rulers and curves (ruler points) works in the same way. In such a case you can't click on two separate points because you don't have two points, so just click twice on the same point.

Cutting in between two rulers

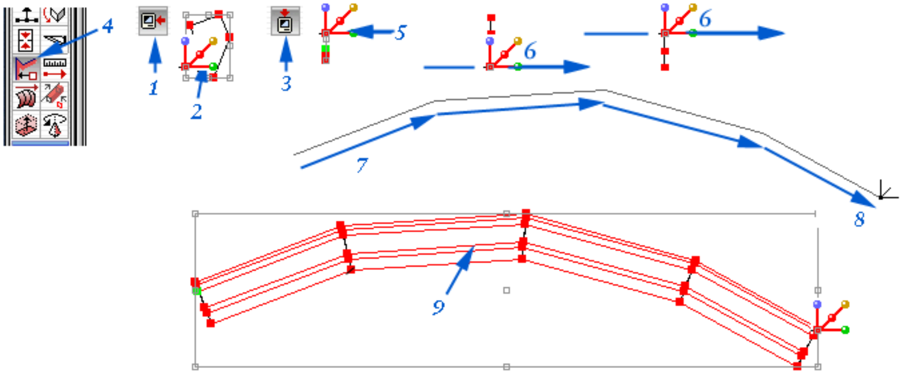
The easiest way is to insert a ***temporary cutting ruler*** or ***set of control points*** (an extra cross section consisting of control points) where you want the cut, and then use the ***Cut*** tool.

Other cutting methods

The ***Split Object -> Along Rulers / Perpendicular to Rulers*** commands in the ***Edit*** menu allows splitting of objects along or perpendicular to the control rulers in one step. Note though that the split takes place on every station along the selected path. One use for such a split is to split a surface into individually editable objects when arranging a cut layout in the Unfold view.

Polygon Extrude tool

The ***Polygon Extrude*** tool is used for ***extracting a cross section*** and ***then extruding it along a polygon*** that you draw. It works like drawing a double-line but you can extrude just about any cross section along the polygon path. Below is a picture showing the steps.



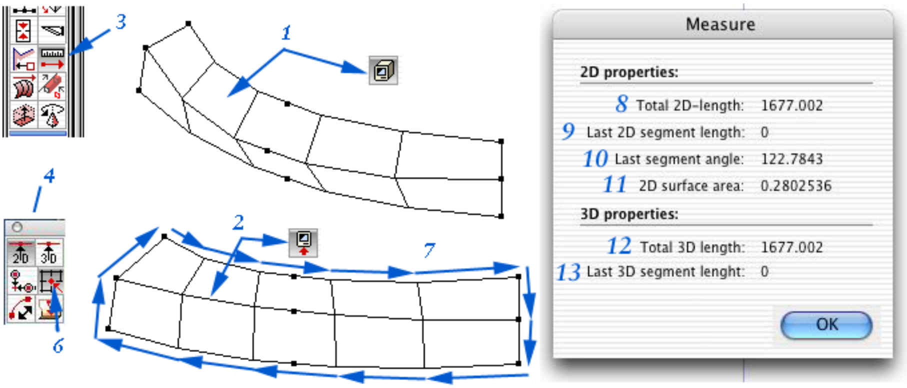
We will extrude a cross section from the ***Top*** view, and need a ***cross section*** being ***vertical*** from that view, that is, it has a ***depth in the direction away / towards you***. Drawing the ***cross section*** seen from the ***Front*** or ***Side*** (Right) views can be used in this case. We have here used the ***Side*** *(1)* (Right) view when drawing the ***cross section*** *(2)* polygon. Select the ***Top*** *(3)* view in the View bar. Select the ***object*** *(5)* by clicking on it with the using the ***Polygon Extrude*** *(4)* tool or by using the ***Selection Arrow*** tool. The location of the Intersection point determines ***offsets*** *(6)* relative to ***the control polygon*** *(7-8)*. Start clicking out the ***extrusion path polygon*** *(7)*. A polygon drawing sequence can be terminated in two ways, by ***double-clicking*** *(8)* on the last point if the shape is open, or by ***clicking once again*** on the ***starting point***. The first alternative was used in this example. Once the sequence is completed the ***new extrusion*** *(9)* occurs.

3D-extrusions

The ***Polygon Extrude*** tool can also be used for generating extrusions having a variable ***depth in the direction away / towards you***. This can be done by using the ***3D-snapping*** option in the ***Constraint*** palette and by ***snapping*** on object points having different height locations (in the direction away / towards you). ***The Cross section*** *(1)*. ***The Extrusion path*** *(2)*. ***The extruded object*** *(3)*.

Measure tool

The measure tool is used as a temporary measuring device for measuring the 2D and 3D length and tilt angles. Below is a picture illustrating the steps.



The illustrated object is a 3D surface shown in the ***Perspective*** *(1)* view. The actual measuring procedure must take place in the standard ***Front*** *(2)*, ***Top*** or ***Side*** (Right) views. Select the ***Measure*** *(3)* tool in the ***Tool*** palette. In this case we want to measure the 3D lengths and using the mesh knots along the edge. To do this, activate the ***2Dsnappin*** *(4)* as well as the ***Snap to Mesh*** *(6)* buttons in the ***Constraint*** *(4-6)* palette. Start ***drawing*** the measuring ***polygon*** in the drawing area by snapping on each ***mesh point*** in consecutive order around the ***surface edge*** *(7)*. Just like the polygon tool, you can terminate the drawing procedure in two ways, by ***double clicking*** on the ***last point***, or by ***clicking once again*** on the ***starting point***, which is the case in this example.

Once the drawing procedure is finished, a dialog occurs showing the measured data. The data shown are as follows. The ***total projected 2D-length*** *(8)* of the drawn line or polygon. The ***projected 2D-length*** of the ***last segment*** *(9)*. The ***2D tilt angle*** *(10)* as seen from the currently selected view. The ***projected 2D-area*** *(11)* of the polygon as seen from the current view. The ***total 3D-length*** *(12)* of the polygon. The ***3D-length*** of the ***last segment*** *(13)*.

Note that the ***area*** assumes a ***closing line*** between the first point used and the last point if the measuring polygon is ***an open polygon***.

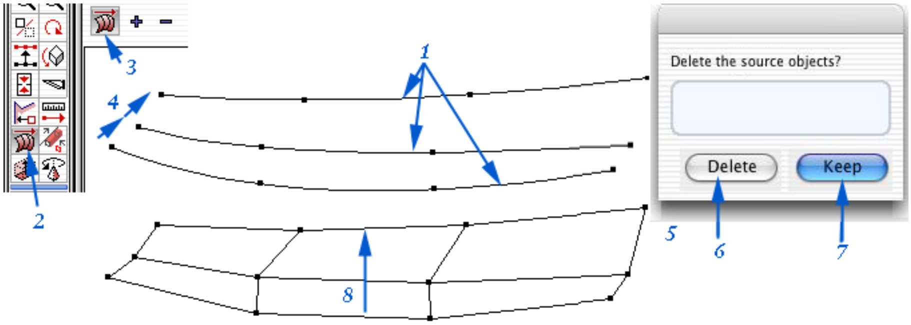
Note that the ***true 3D-area*** of the surface can be calculated using the ***Surface Properties*** commands in the ***Tool menu***.

Note that the ***2D- and 3D-lengths*** are not the same here because the ***3D-snapping*** made the measuring point three-dimensional, that is, it uses the coordinates in the direction ***away / towards you*** as well.

The Measure tool works in the Front, Top, Side (Right) and the Unfold views.

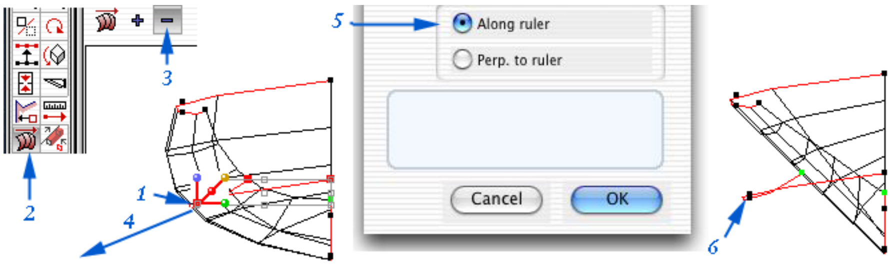
Loft tool

The Loft tool is used for merging line / curve / ruler objects into a single surface object. It can also be used for removing rulers from a surface object into a separate object, and insert-lofting an extra ruler into an already existing surface object. Below is a picture showing the steps.



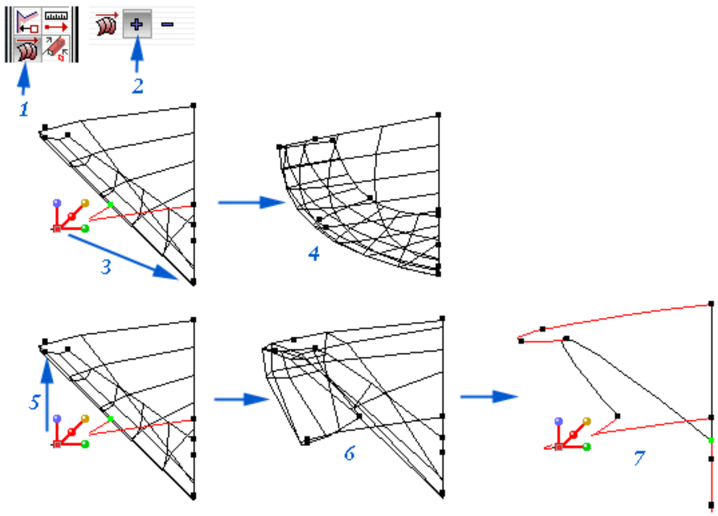
Building a surface

Start by drawing for example three ***control rulers*** *(1)* using the Polygon tool found in the blue section of the tool palette. ***Curves*** (a series of ruler points) can also be used. Select the ***Loft*** *(2)* tool in the ***Tool*** palette. Select the ***Build Surface*** *(3)* mode in the ***Mode*** bar. ***Click*** *(4)* on ***one*** of the ***control points in each of the objects*** to be used for the surface ***in consecutive order*** starting from one side. ***Click twice*** on ***the last curve point*** (***or beside*** any of the curves) to ***finish*** the sequence. A ***dialog*** *(5)* occurs where you specify if you want to ***keep*** *(7)* the original lofting objects ***or not*** *(6)*. The default setting is to keep the original curves but in reality it is often best to delete them unless you have an explicit use for them later on. Once this selection has been made the final result is being ***displayed*** *(8)*. This surface may look rough but that can easily be fixed by ***increasing the surface resolution*** using the ***Object Info*** palette.



Removing a ruler or curve

The ***removing*** and ***inserting*** procedures work in a fairly similar way. To remove for example the center ruler from the surface we just created, just select one relevant control point with the ***Selection Arrow*** *(1)*. Select the ***Loft*** *(2)* tool and then the ***Delete*** *(3)* (minus) button in the mode bar. ***Click*** on the ***selected*** point, ***drag*** and then ***click*** again ***beside*** *(4)* the surface. A ***dialog*** *(5)* occurs where you ***specify*** if you want to remove the selected ***control ruler*** or a ***curve*** being perpendicular to the control rulers. Click on the relevant direction and the selection will be ***removed*** *(6)* from the surface.



Inserting a ruler or curve

Inserting a ***curve*** (a series of ruler starting points) or ***control ruler*** is done in the same way except that you use the ***Insert*** *(2)* (plus) button in the ***mode*** bar and that you draw the ***connection line from the insertion object to the receiving object***.

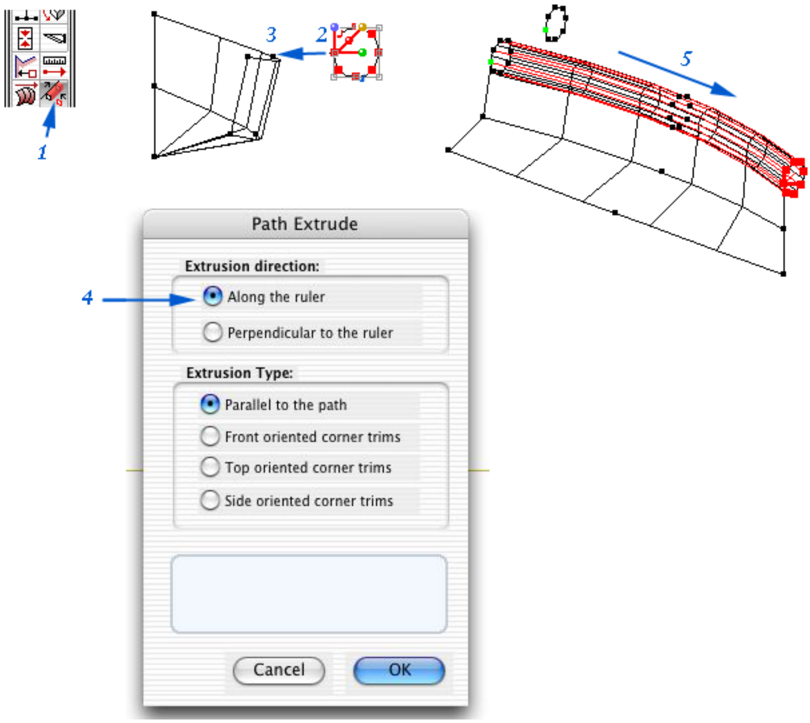
It is important to realize that the ***inserted curve / ruler*** is placed ***above the ruler you click on in the receiving surface***, counted from the ***green starting point***.

In the above mentioned example you will get back to the shape you had before if you drag the ***insertion ruler*** to the ***ruler with the green dot*** *(3-4)*, because ***the inserted ruler*** will then be ***located immediately above the ruler having the green dot*** *(5-6)*.

If you on the other hand drag the ***insertion ruler to the upper ruler***, it will be ***inserted as the third curve in the list*** and this will result in a very different shape. The last picture *(7)* shows the surface as a contour and here you clearly see that the intermediate curve in the *(3-4)* example is located as the last ruler in the shape, referenced from the green point dot.

Path Extrude tool

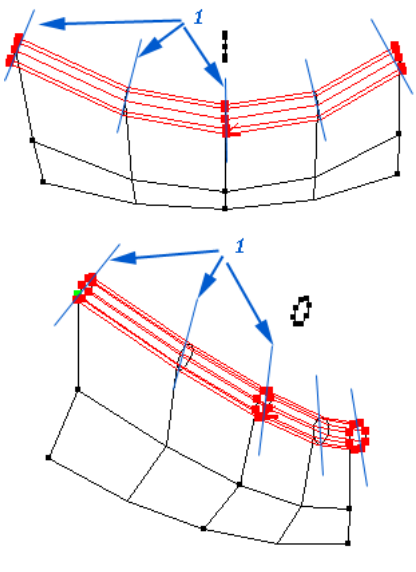
The *Path Extrude* tool is used for extruding cross sections along a given ruler or curve. The *path* can be an ***individual control ruler*** or ***curve*** (a series of ruler starting points), but it can also be an ***element in a surface, along the edge*** as well as somewhere ***in the middle***. Below is a picture showing a ***path extrusion*** along the ***edge of a surface*** where the ***edge ruler*** *(3)* is used as the ***extrusion path***.



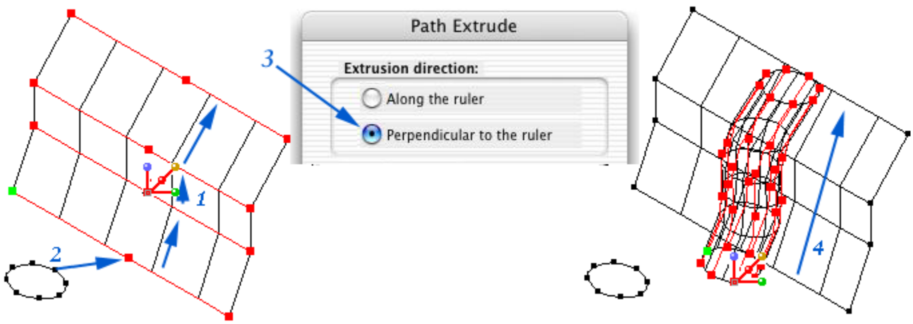
Extruding along a ruler

Start by drawing a ***cross section*** *(2)*, for example by using the tools in the blue section of the *Tool* palette. Be sure to draw in a view roughly perpendicular to the ***extrusion path*** *(3 & 5)*. The ***cross section*** *(2)* object can be a ***ruler*** or a ***curve*** (a series of ruler points) but ***not a surface***. Select the *Path Extrude (1)* tool in the *Tool* palette. Draw a connection line between the ***connection point*** of the ***cross section*** and the ***extrusion path*** *(2->3)*. Note that it is important to choose the relevant cross section point (2) as it controls where the extrusion is placed relative to the path. In the example above the leftmost side was used and that places the entire extrusion to the right of the path curve. If the rightmost point had been selected the extrusion would placed to the left of the extrusion. The same applies in the up /down direction.

A dialog occurs where you specify that you want to ***extrude*** the cross section ***along the path ruler*** *(4)*. The extrusion takes place when you click on the OK button in the dialog box. The ***result*** *(5)* is here shown in a slightly rotated view.



The ***Extrusion type*** options control the behavior of the extrusion. The ***parallel option*** *(2)* generates a simple offset with no corner adjustments. The other three options (1) essentially behave as when drawing a double-line or when using the Polygon Extrude tool. They automatically rotate and expand the joints so that the cross section remains constant relative to the view used. The Front option behaves as if you where drawing a double line in the Front view and so on. This is why there are three view options.

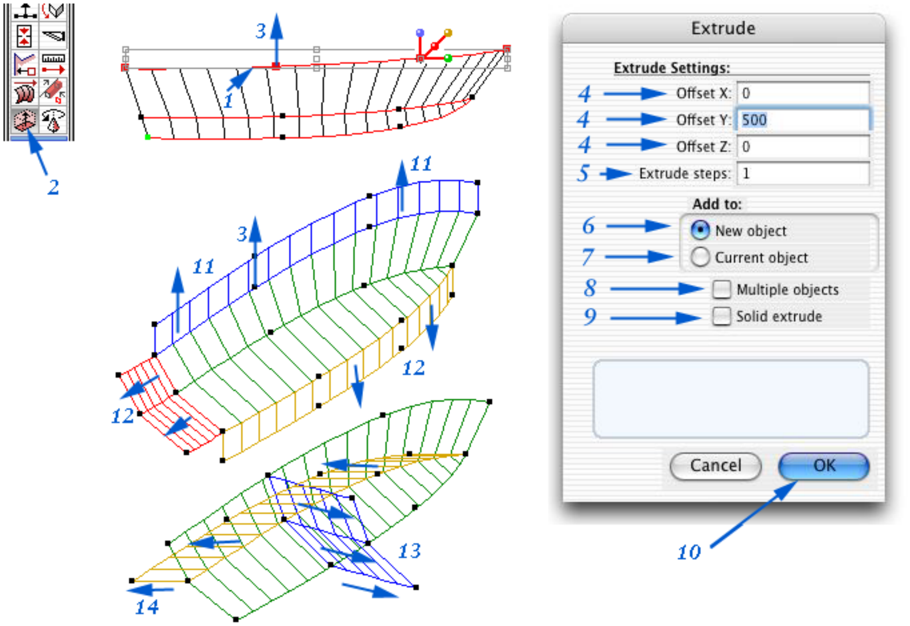


Extruding perpendicular to a ruler

Above is a picture showing a ***path extrusion perpendicular*** to ***the control rulers*** and placed at the same ***control point index*** *(1)* on each control ruler (in the middle in this example). The procedure is the same as before. Drag ***the cross section to the path*** *(2)*, select the ***direction*** *(3)* and the extrusion takes place *(4)*.

Extrude tool

The ***Extrude*** tool can be used both for adding more ***control points within*** the currently selected ***object,*** and for adding ***new objects*** based on the selected object or objects.



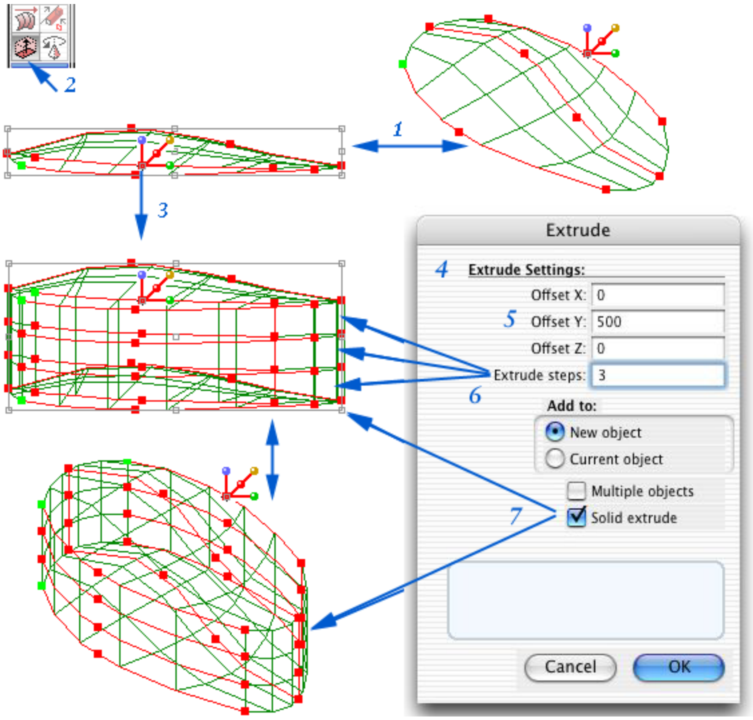
Extruding a new surface

Above is an example showing how to extrude a new connecting surface from the edge of a ***selected object*** *(3)*. Start by selecting at least two points along the ***edge*** *(1)*. Selecting ***two ruler points*** in a ruler indicates that you want to ***base the extrusion*** on ***the control ruler***. Selecting ***two control points*** being in ***different rulers*** but having the same point index indicates an ***extrusion*** being ***perpendicular to the control rulers***. Select the ***Extrude*** *(2)* tool in the tool palette. Click on one of the selected control rulers and ***drag*** *(3)* and then ***click*** to finish the extrusion line. A dialog occurs. The ***Offset*** *(4)* fields specify the offsets in the X, Y and Z directions. The ***Extrude Steps*** *(5)* indicates the number of rulers to be added in the extrusion. ***New Surface*** *(6)* and ***Current Surface*** *(7)* indicates whether the new items are to be ***added within the current object*** or if a ***new object is to be added***. ***Multiple Surfaces*** *(8)* allows simultaneous extrusion of several objects. ***Solid Extrude*** *(9)* adds a thickness and edge surfaces to the currently selected surface. Click ***OK*** *(10)* to add the ***new surface*** *(11)*.

Note that the ***Extrude*** tool allow ***extrusions from any edge side*** (12), ***internally*** in the middle of a ***surface along a ruler*** *(14), **and internally / perpendicular to the control rulers*** *(13)*.

Solid Extrude

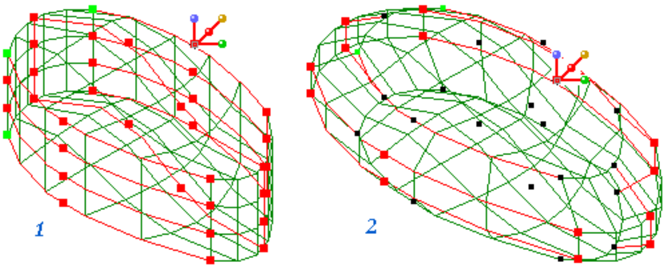
The *Solid Extrude (9)* option in the ***Extrude Dialog*** allows you ***to add a thickness*** to a surface. Below is a picture showing the steps.

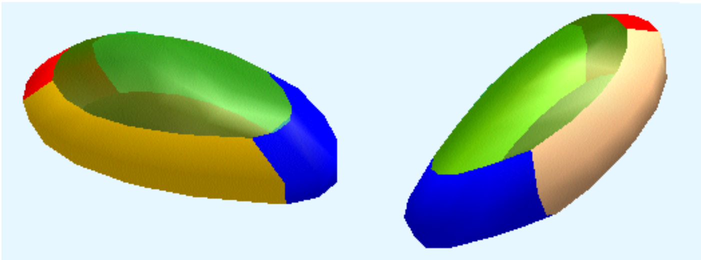


Select a ***surface*** *(1)* using the ***Selection Arrow*** tool in the ***Tool*** palette. The picture shows it from the ***side***, which is ***the normal operational view*** for such an operation, and in a perspective view to clarify the shape. Select the ***Extrude*** *(2)* tool in the ***Tool*** palette. Click on the surface, drag the ***extrusion line*** to the ***approximate extrusion depth*** *(3)* and then ***click*** to finish the sequence. The ***Extrude dialog*** *(4)* occurs where you adjust ***the extrusion distance*** *(5)*, specify the number of ***extrusion steps*** *(6)*, and that you want ***a solid extrusion*** *(7)*. Click OK and you have added a thickness to the surface as seen to the bottom left.

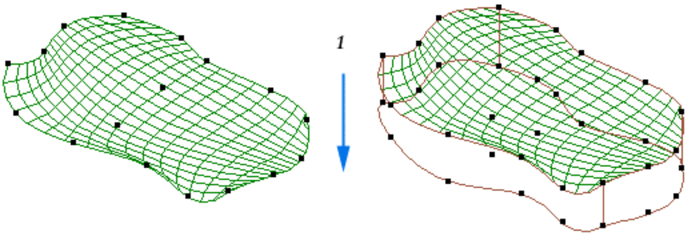
The ***Solid Extrusion*** tool essentially adds ***four side walls*** and then ***copies*** the ***master surface*** to form the ***opposite side*** of the object. The ***extrusion*** consists of ***six objects*** and is therefore ***grouped by default.*** To ***edit*** the object, select the ***Ungroup*** command in the ***Edit*** menu. You can also use the ***Ignore Groups*** command in the ***Edit*** menu to ***temporarily disable*** the ***group*** behavior and allow editing without ungrouping the group. Select ***Ignore Group*** menu command again ***to reset*** the ***Ignore Group*** mode.

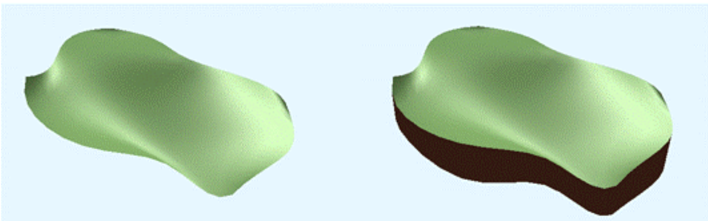
The default is to use ***one extrusion step*** on the sides to create a ***straight connection*** between the upper and lower sides. If the sides are intended to have ***curved sides*** it is better to ***add some intermediate steps*** such as shown below.

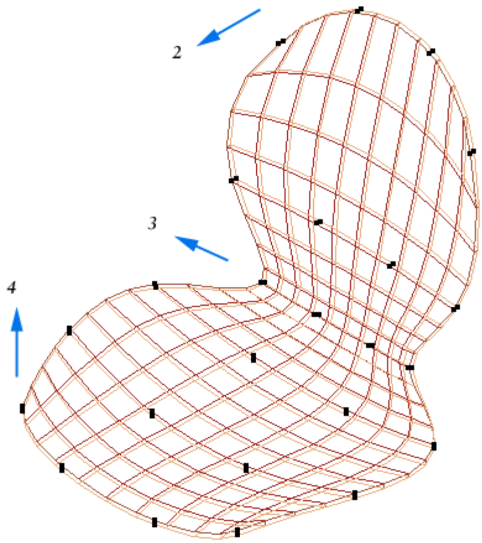




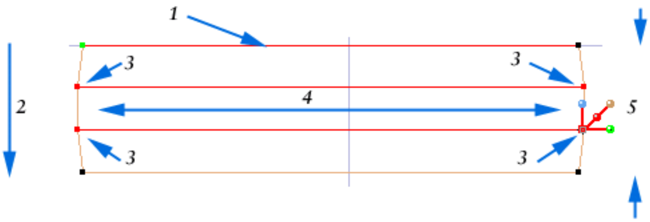
Below are a few examples where this technique has been used. The terrain model originates from a single ***surface extruded downward*** in a ***single step***. The bottom is made flat by, choosing the ***Ignore Groups mode***, ***selecting all control points along the bottom***, click on the ***Y field*** above the drawing window, and ***add "=" before the coordinate***. Press ***Enter***. That's all it takes to generate a flat underside for the extrusion.

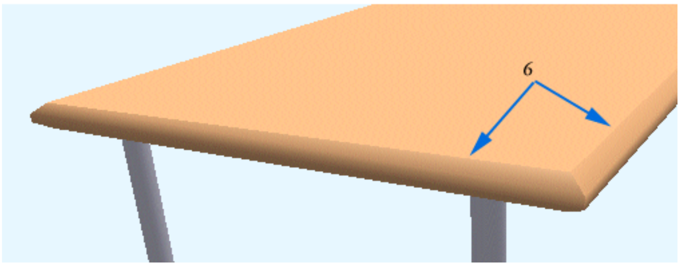




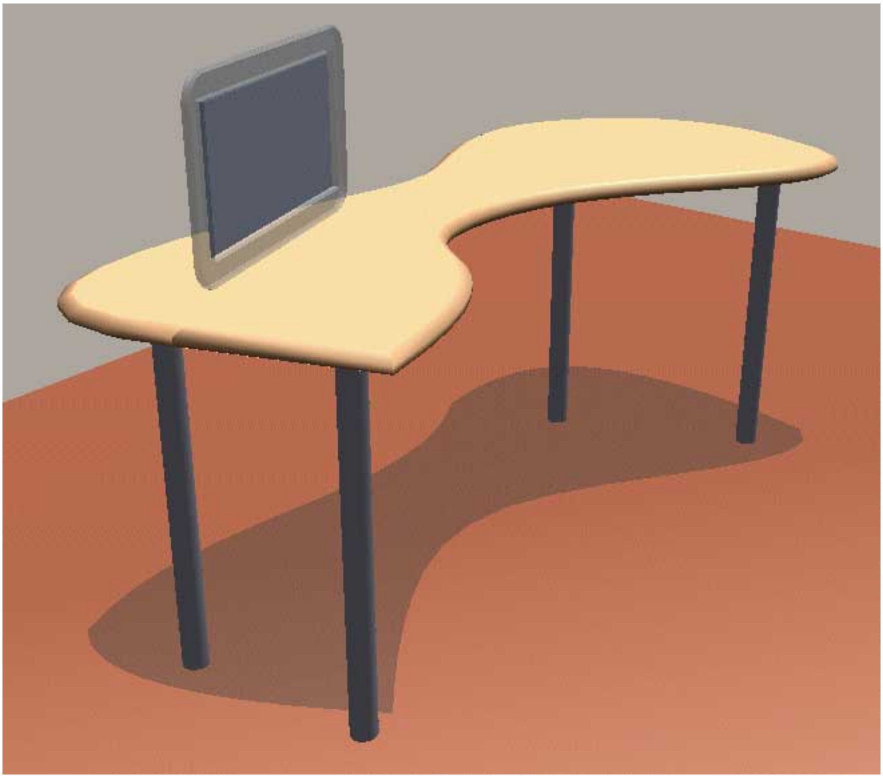


The chair is created in a similar way. Exaggerate the extrusion, drag ***the vertical (2) points*** back to the original location and then ***nudge them forward***, in this case about 6 mm. Move the ***bottom parts (4) upward*** and the ***joint parts both (3) forward and up***.





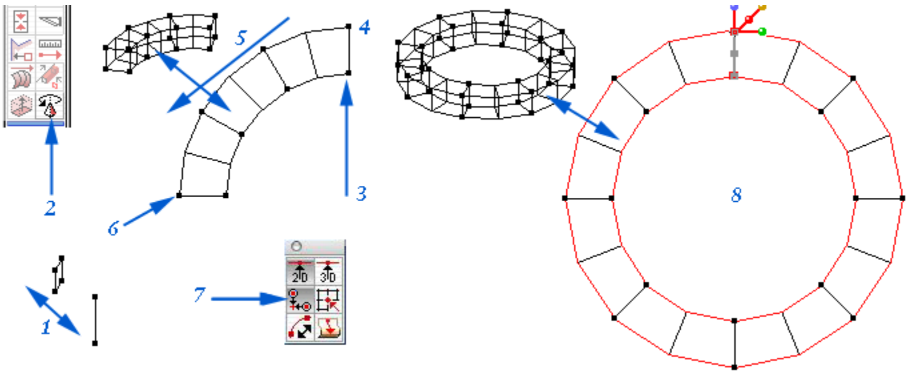
A third example shows a simple way to add ***rounded edges*** to a table surface. A simple rectangular surface drawn in the ***Top*** view is the basis for the extrusion (1). The surface is then ***extruded*** seen from the ***Front*** view, using the ***Solid Extrusion*** method and adding ***three new extrusion steps (2)***. Note that the extrusion is much thicker than the final thickness of the table surface. This makes it easier to select objects and can easily be fixed later in the process. The eight intermediate corner points are selected using the ***Selection Arrow*** (***Mesh*** mode) (***3***). Select the ***Reshape*** command in the ***Edit*** menu. Add 30 mm in the X and Z directions and scale for the center point (***4***). Select everything and use the reshape command to change the height to 30 mm (***5***). The final result can be seen in the image as a rendered image (***6***). You may need to change the ***Skin Type*** setting in the ***Object Info*** palette for two sides to get the smoothing right when rendering.



This method works fairly well on non-rectangular shapes too, though it may need some minor editing/nudging to get a more or less perfect result as seen above.

Sweep tool

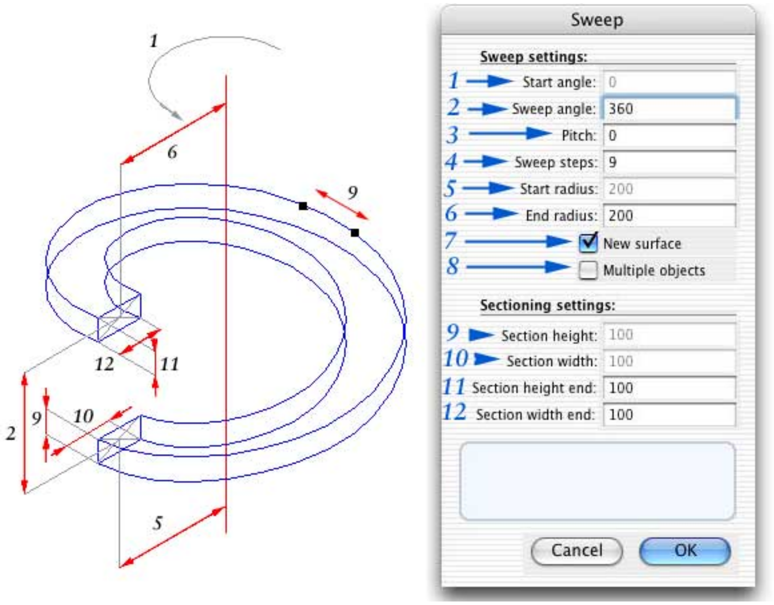
The Sweep tool is used for creating surfaces based on a cross section. The tool can be used both for ***adding more control points within objects*** and for ***new objects*** based on the selected ***cross sectional object or objects***. Below is a picture showing the steps.



Start by drawing a ***cross section*** *(1)*. It should be oriented ***perpendicular*** to the ***sweep*** direction. Change view to where the cross section looks like a ***line*** *(1)*, which is the correct direction using the ***Sweep*** tool. Select it using the **Selection Arrow** tool in the ***Tool*** palette. Select the ***Sweep*** *(2)* tool in the ***Tool*** palette. ***Click*** at the center of ***rotation*** *(3)*, draw the ***first sweep angle arm*** and then ***click*** *(4)* to place it. Rotate the ***lever arm*** *(5)* to a ***second location*** and ***click*** *(6)* again. A ***sweep dialog*** occurs, in which you just ***click OK*** for now. The ***swept object*** *(8)* is now generated.

Note that it can be quite useful to either place a ***temporary center-point*** in order to have something to ***snap*** on when using the ***Sweep*** command. In some cases it can also be useful to turn on the ***Alignment Indicator*** *(7)* found in the ***Constraint*** palette. These methods can provide useful assistance.

When drawing a completely ***circular object*** *(8)* you simply ***click twice*** *(4, 6)* on one and the same point. In such a case TouchCAD assumes that you want to draw a ***circular*** shape and not a zero-sweep object.



We have previously covered the basic use of the *Sweep* tool. The options in the *Sweep dialog* are as follow:

Start angle (1). This field displays the ***angle*** between the ***first center-point*** click and the ***second click***. This is a fixed setting.

Sweep angle (2). This field displays the ***angle*** between the ***first and second lines***. Note that if you ***click twice*** on the ***second point*** it is assumed that you want to draw a full ***360-degree circle*** rather than a zero-degree arc. You can also insert an ***angel bigger*** than ***360 degrees***, for example 720 or 1440 degrees. This only makes sense if you include some sort of ***deformation along the path*** such as a ***pitch*** or ***a radius transformation***.

Pitch (3). The ***Pitch*** option allows you to generate a ***spiral shape*** such as a coil spring. The value defines the ***total height offset*** of the sweep.

Sweep steps (4). TouchCAD automatically calculates ***how many control points*** you need to control the ***specified sweep surface***. The ***Sweep steps*** option can in most cases be left untouched. In some cases it is however practical to use a different setting, for example to be able to control a more complex shape.

Start Radius (5). Defines the ***start radius of the sweep*** (the ***distance*** from ***the center point*** to the ***center of the cross section***). This is not an editable field and it is only used for reference.

End Radius (6). Defines the ***radius at the end of the sweep***. A value being ***higher*** than in the ***start radius gradually increases the radius*** along the sweep path. A ***lower value decreases*** the sweep ***radius*** along the sweep path.

New Surface (7). If the *New Surface* option is selected, TouchCAD creates ***a new separate surface*** from the selected control points. ***If not***, the ***sweep is added*** to the ***currently selected surface***. It may sound strange, but you can for example add a ***spiral*** or and ***S-curve*** in the middle of a straight extrusion.

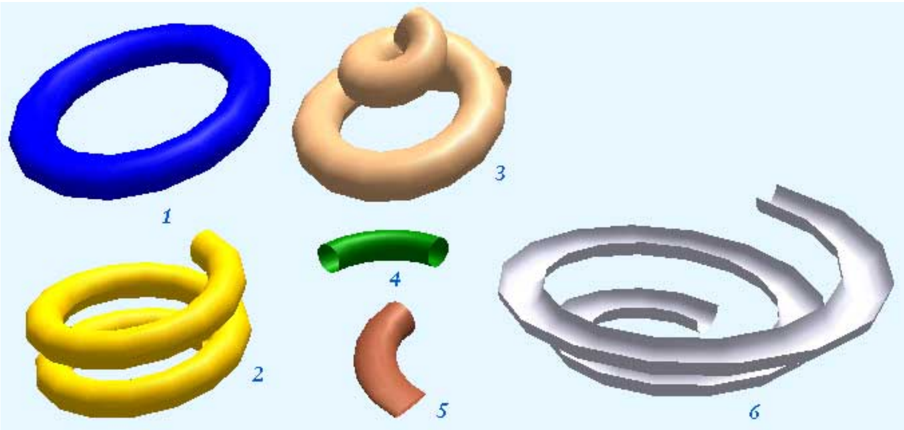
Multiple surfaces (8). Allows you to simultaneously create ***several sweeps*** from control points placed in ***different objects***.

Height and width (9-10). Defines the ***height*** and ***width*** of the ***cross section*** at the ***start of the sweep***. Not editable.

Height and Width (11-12) of the ***cross section*** at the ***end of the sweep***. These features allow you to ***gradually transform*** the ***cross section*** dimensions ***along the sweep path***.

Sweep examples

Below is a picture showing some examples of sweep shapes.

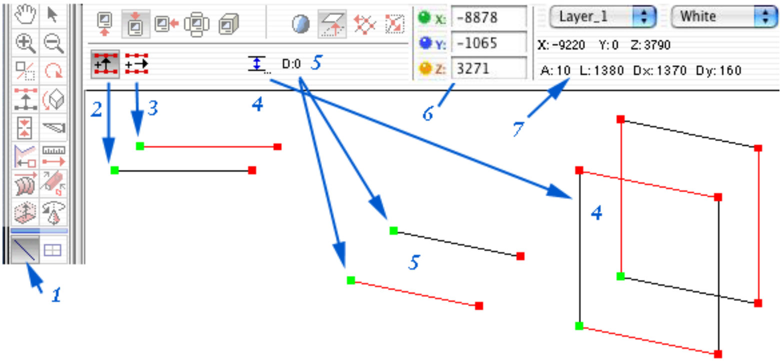


The first example shows a basic *360-degree sweep (1)* founded on ***a circular cross section***. The second example shows a *720-degree sweep (2)* that has a pitch. The third example shows a *720-degree sweep (3)* with a ***pitch*** and the ***sweep radius decreases along the sweep path***. The fourth example shows a simple *90-degree sweep (4)*. The fifth example is also a *90-degree sweep (5)* but where the ***cross section transforms*** from a circular section into a fairly flat oval. The sixth example illustrates that nearly any ***cross-section*** can be used, in this case an ***arc*** with a *720-degree sweep (6)*, a ***pitch*** and wherein ***the sweep radius increases*** along the sweep path.

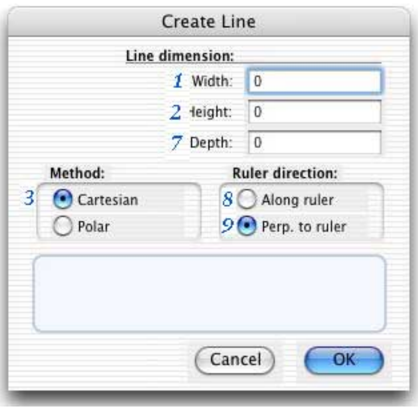
These examples illustrate the wealth of options in just the basic ***Sweep*** tool. You can of course use any combination of these settings. Note that the ***sweep*** objects are still the same ***math-curve based surfaces*** as in the rest of the program. They therefore allow you to ***extract*** even ***more complex shapes based on these sweeps***.

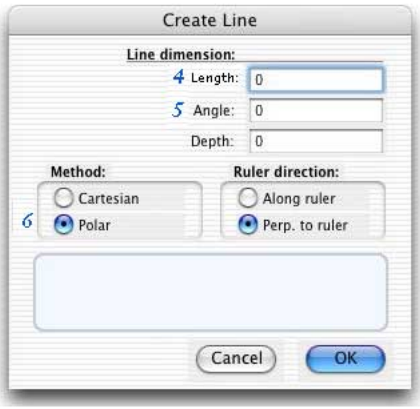
Line tool

The *Line (1)* tool in the Blue section of the *Tool* palette is used for drawing basic 2D-lines. It can also generate ***surfaces*** if the ***depth setting*** is set to a value ***higher than zero***. Below is a picture showing the steps.



A line is drawn using the ***Line*** *(1)* tool in the ***Tool*** palette The Mode bar provides two drawing options, ***Curves*** *(2)* (a series of ruler starting points) and ***Control Rulers*** *(3)*. The ***Depth*** *(4)* button displays a depth dialog where the depth is set. The depth controls if you want the line to be ***extruded*** *(4)* or ***not*** *(5)*. A ***depth*** of ***zero gives no extrusion*** whereas a depth ***higher than zero*** adds an ***extrusion depth*** to the line. The current ***depth setting*** *(5)* is displayed in the mode bar. The ***location*** of the line in the direction ***away /towards*** you is controlled with the ***three coordinate edit fields*** *(6)* above the drawing area. If you, for example, are in the ***Top view*** while ***drawing*** the line, and with no 3D snapping activated, the line ends up at the current ***Y coordinate field*** *(6)* ***setting***. TouchCAD also displays various additional ***dimensional data*** *(7)* while drawing a line.





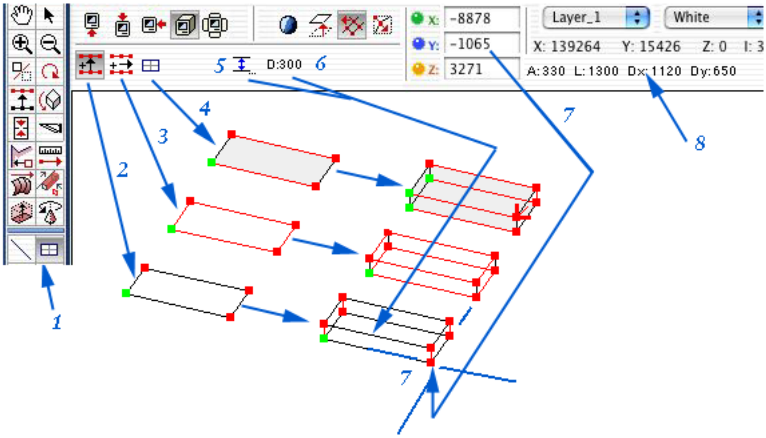
You can also draw a line numerically by ***double clicking*** on the ***Line*** tool, or by selecting the ***Line command*** in the ***Tool*** menu. A dialog is shown in both these cases, where you can enter the size of the line.

The ***Cartesian*** *(3)* ***mode*** defines a line using a ***height*** *(1)* and ***width*** *(2)* whereas the ***Polar*** *(6)* ***mode*** defines it by means of a ***diagonal length*** *(4)* and ***angle*** *(5)*.

The ***Depth*** *(7)* is the same as described on the previous page and so are the line types, ***Along*** *(8)* and ***Perpendicular*** *(9)* ***to the ruler***.

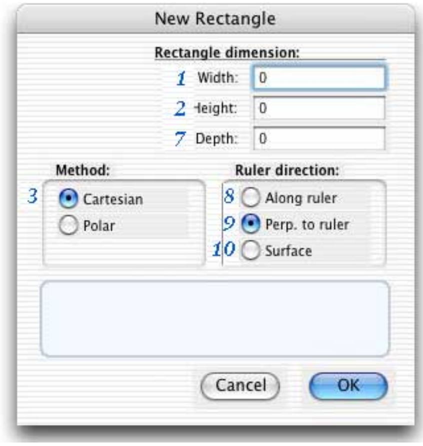
Rectangle tool

The ***Rectangle** (1)* tool found in the Blue section of the ***Tool*** palette is used for drawing basic ***2D-rectangles***. It can also be used for generation ***cubes and cubical shapes*** if the ***depth setting*** is set to a ***value higher than zero***. Below is a picture showing the steps.



A rectangle is drawn using the ***Rectangle** (1)* tool in the ***Tool*** palette. The ***Mode*** bar provides three drawing options, ***Curves** (2)* (a series of ruler starting points), ***Control Rulers** (3)* and ***surface** (4)* where a surface object consisting of ***two rulers*** each having ***two control points***. The ***Depth** (5)* button displays a ***depth dialog***, in which the depth is set. The ***Depth** (5-6)* controls if you want the rectangle to be ***extruded*** or ***not***. A ***depth of zero*** gives ***no extrusion*** whereas a depth ***higher than zero adds an extrusion depth*** to the rectangle. The current ***depth setting** (6)* is displayed in the mode bar. The ***location*** of the rectangle in the direction ***away /towards*** you is controlled with the ***three coordinate edit fields** (7)* above the drawing area. If you, for example, are in the ***Top view*** while ***drawing*** the rectangle, and with no 3D snapping activated, the rectangle ends up at the current ***Y coordinate field** (7) **setting***. TouchCAD also displays various additional ***dimensional data** (8)* while drawing a rectangle.

Note that ***surface rectangles*** drawn ***with a depth creates a group consisting of six surfaces***. The ***curve and ruler*** based ***objects*** consists of a ***single surface*** and does not have a bottom or top surface. To ***edit*** individual control points in the ***group***, choose the ***Ignore Groups*** mode command in the ***Edit*** menu or use the ***Ungroup*** command in the ***Edit*** menu.



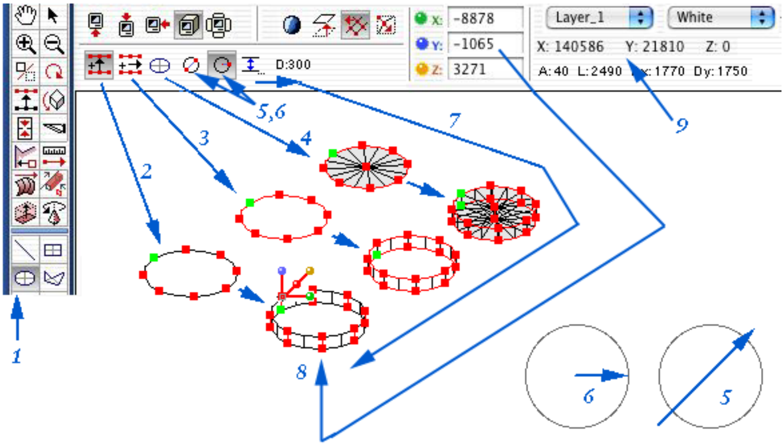
You can also draw a rectangle numerically by ***double clicking*** on the ***Rectangle*** tool, or by selecting the ***Rectangle command*** in the ***Tool*** menu. A dialog is shown in both these cases, wherein you can enter the size of the rectangle.

The ***Cartesian*** *(3)* ***mode*** defines a rectangle using a ***height*** *(2)* and ***width*** *(1)* whereas the ***Polar*** *(4)* ***mode*** defines it by means of a ***diagonal length*** *(5)* and ***angle*** *(6)*.

The ***Depth*** *(7)* is the same as described on the previous page and so are the rectangle types, ***Along*** *(8)*, ***Perpendicular*** *(9)* ***to the ruler,*** and ***Surface*** *(10)*.

Circle / Ellipse tool

The *Circle / Ellipse (1)* tool in the Blue section of the ***Tool*** palette draws basic ***2D circles*** and ***ellipses***. It can also generate ***cylinders*** and ***cylindrical shapes*** if the ***depth setting*** is set to a value ***higher than zero***. Below is a picture showing the steps.

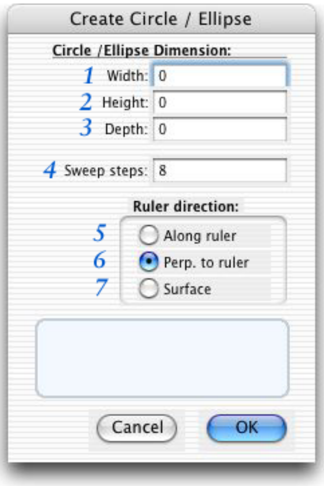


A Circle / Ellipse is drawn using the *Circle / Ellipse (1)* tool in the ***Tool*** palette The Mode bar provides three ***drawing options***, *Curves (2)* (a series of ruler starting points), *Control Rulers (3)* and *Surface (4)* where a surface object consists ***of two control rulers*** (one in the middle).

Two drawing methods can be used, ***diagonally*** *(5)* and as a ***radius*** *(6)* from the center-point. The ***diagonal*** *(5)* method allows you to draw ***ellipses***, but you can also draw ***circles by*** pressing the ***Shift*** key while dragging.

The ***Depth*** *(7)* button displays a ***depth dialog*** where the depth is set. The ***Depth*** *(7)* controls if you want the ***circle / ellipse*** to be ***extruded*** or ***not***. A ***depth of zero gives no extrusion*** whereas a depth ***higher than zero*** adds an ***extrusion depth*** to the circle / ellipse. The current ***depth setting*** *(7)* is displayed in the mode bar. The ***location*** of the Circle / Ellipse in the direction ***away /towards*** you is controlled with the ***three coordinate edit fields*** *(8)* above the drawing area. If you, for example, are in the ***Top view*** while ***drawing*** the Circle / Ellipse, and with no 3D snapping activated, the circle /ellipse ends up at the current ***Y coordinate field*** *(8)* ***setting***. TouchCAD also displays various additional ***dimensional data*** *(9)* while drawing a circle / ellipse.

Note that ***surface circles / ellipses*** drawn ***with a depth build a group consisting of several surfaces***. ***Curve and ruler*** based ***objects*** consists of a ***single surface*** and does not have a bottom or top surface. To ***edit*** individual control points in the ***group***, choose the ***Ignore Groups*** mode command in the ***Edit*** menu or use the ***Ungroup*** command in the ***Edit*** menu.



You can also draw a ***Circle / Ellipse*** numerically by ***double clicking*** on the ***Circle / Ellipse*** tool, or by selecting the ***Circle / Ellipse command*** in the ***Tool*** menu. A dialog is shown in both cases, wherein you can enter the size of the Circle / Ellipse.

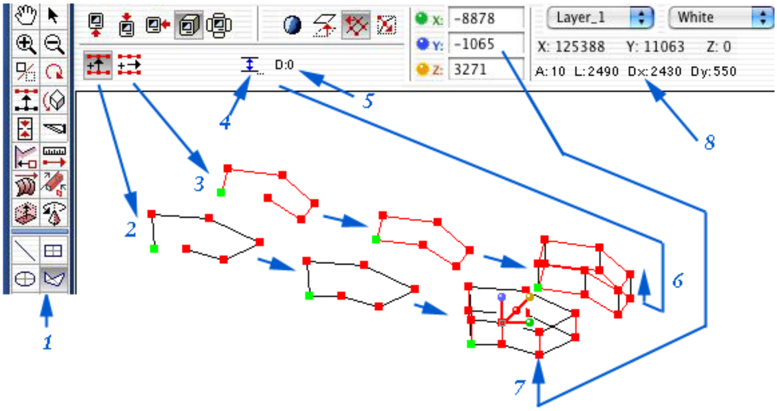
The ***width*** *(1)* and ***height*** *(2)* fields define the basic dimensions.

The ***Sweep Steps field*** *(4)* defines the number of sweep steps used along the sweep path.

The ***Depth*** *(3)* is the same as described on the previous page and so are the Circle / Ellipse types, ***Along*** *(5),* ***Perpendicular*** *(6)* ***to the ruler,*** and ***Surface*** *(7).*

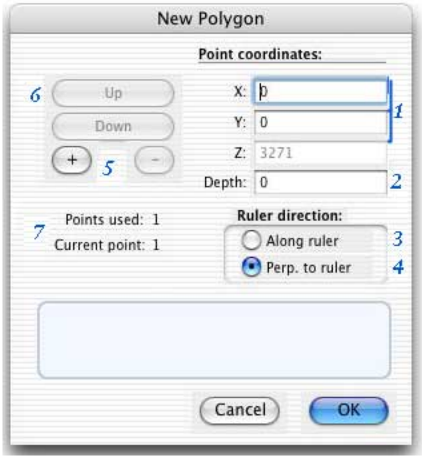
Polygon tool

The ***Polygon (1)*** tool in the Blue section of the ***Tool*** palette draws basic ***2D polygons***. It can also generate ***surfaces*** if the ***depth*** is set to a value ***higher than zero***. Below is a picture showing the steps.



A polygon is drawn using the ***Polygon (1)*** tool in the ***Tool*** palette. The ***Mode*** bar provides two drawing options, ***Curves (2)*** (a series of ruler starting points) and ***Control Rulers (3)***. The ***drawing sequence*** can be ***terminated*** in two ways, by ***clicking*** on the ***starting point***, or by ***clicking twice*** on the ***last point***. The ***Depth (4)*** button displays a depth dialog where the ***depth (5)*** is set. The ***depth*** controls if you want the polygon to be ***extruded (6)*** or ***not (2-3)***. A ***depth of zero*** gives ***no extrusion*** whereas a ***depth higher than zero*** adds an ***extrusion*** depth to the polygon. The current ***depth setting (5)*** is displayed in the mode bar. The ***location*** of the polygon in the direction ***away /towards*** you is controlled with the ***three coordinate edit fields (7)*** above the drawing area. If you, for example, are in the ***Top view*** while ***drawing*** the polygon, and with no 3D snapping activated, the polygon ends up at the current ***Y coordinate field (7) setting***. TouchCAD also displays various additional ***dimensional data (8)*** while drawing a polygon.

The ***Polygon*** tool sets the ***object resolution*** to the same resolution as the ***number of control points used***. The ***resolution*** can however be ***changed*** using the settings under the ***Properties*** tab in the ***Object Info*** palette. The same applies to the ***smoothing properties,*** which by default are set to straight in both directions. It enables you to ***convert a polygon*** from having ***straight lines*** between the ***control points*** into a ***smooth looking curve***, and wherein the degree of smoothing it is controlled numerically in a very accurate and flexible way.



You can also draw a polygon numerically by ***double clicking*** on the ***Polygon*** tool, or by selecting the ***Polygon command*** in the ***Tool menu***. A dialog is shown in both cases, where you can enter the size of the polygon.

The ***Coordinate*** *(1)* fields are used for editing the ***currently selected control point coordinates***.

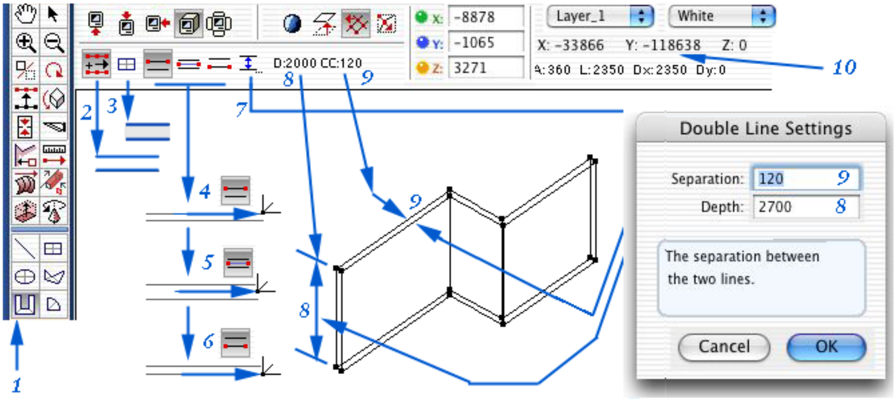
The ***Depth*** *(2)* is the same as described on the previous pages and so are the polygon types, ***Along*** *(3)* and ***Perpendicular*** *(4)* ***to the ruler***.

The ***Plus and Minus*** *(5)* are used for adding and deleting control points respectively. ***The Up and Down*** *(6)* buttons control wherein the ***control point list*** new objects are ***being added or deleted.***

The ***Points used*** and ***Current point*** *(7)* lead texts specify where you are in the control point list.

Double-line tool

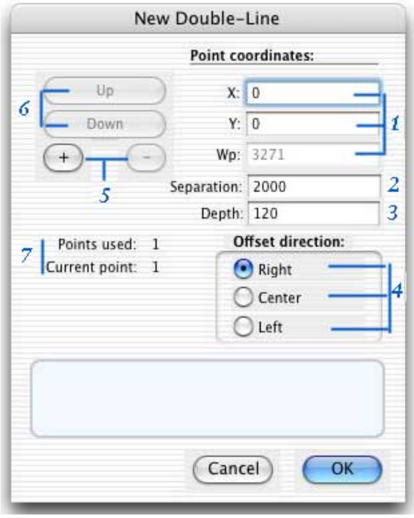
The ***Double-line (1)*** tool in the Blue section of the ***Tool*** palette draws basic ***2D double-lines***. It can also generate ***wall-style objects*** if the ***depth*** setting is set to a value ***higher than zero***. Below is a picture showing the steps.



In a double-line **drawn** using the ***Double-line (1)*** tool in the ***Tool*** palette the ***Mode*** bar provides two drawing options, ***Control Rulers (2)*** and ***Surface (3)***. The ***drawing sequence*** can be ***terminated*** in two ways, by ***clicking*** on the ***start point***, or by ***clicking twice*** on the ***current point***. The drawing sequence can be ***right*** *(5)*, ***center*** *(5)* or ***left*** *(6)* ***oriented***. The ***Depth (7) button*** displays a depth ***dialog*** *(7)*, wherein the ***depth*** *(8)* and ***distance*** *(9)* ***between the lines*** are set. The ***depth*** controls whether you want the double-line to be ***extruded*** *(6)* or ***not*** *(2-3)*. A ***depth equals zero*** gives ***no extrusion*** whereas a ***depth higher than zero*** adds an ***extrusion*** depth to the double-line. The current ***depth setting*** *(9)* is displayed in the mode bar. The ***location*** of the double-line in the direction ***away /towards*** you is controlled by the ***three coordinate edit fields*** above the drawing area. If you, for example, are in the ***Top view*** while ***drawing*** the double-line, and with no 3D snapping activated, the double-line ends up at the current ***Y coordinate field setting***. TouchCAD also displays various additional ***dimensional data*** *(10)* while drawing a double-line.

The ***double-line*** tool sets the resolution to the same resolution as the number of ***control points*** used. The ***resolution*** can however be ***changed*** using the settings under the ***Properties*** tab in the ***Object Info*** palette. The same applies to the ***smoothing properties***.

Note that ***double-lines*** being drawn ***with a depth*** and based on the ***Surface*** method ***form a group consisting of many surfaces.*** R***uler-based objects*** consist of a ***single surface***, and does not have a bottom or top surface. To ***edit*** individual control points in the ***group***, choose the ***Ignore Groups*** mode command in the ***Edit*** menu or use the ***Ungroup*** command in the ***Edit*** menu before editing.



You can also draw a double-line numerically by ***double clicking*** on the ***Double-line*** tool, or by selecting the ***double-line command*** in the ***Tool menu***. A dialog is shown in either case, wherein you can enter the elements of the double-line.

The ***Coordinate*** *(1)* fields are used for editing the ***currently selected control point*** coordinates.

The ***Separation*** *(2)* field specifies the ***distance between*** the two lines.

The ***Depth*** *(3)* is the ***depth*** of the double-line, that is, in the ***direction away / towards*** you.

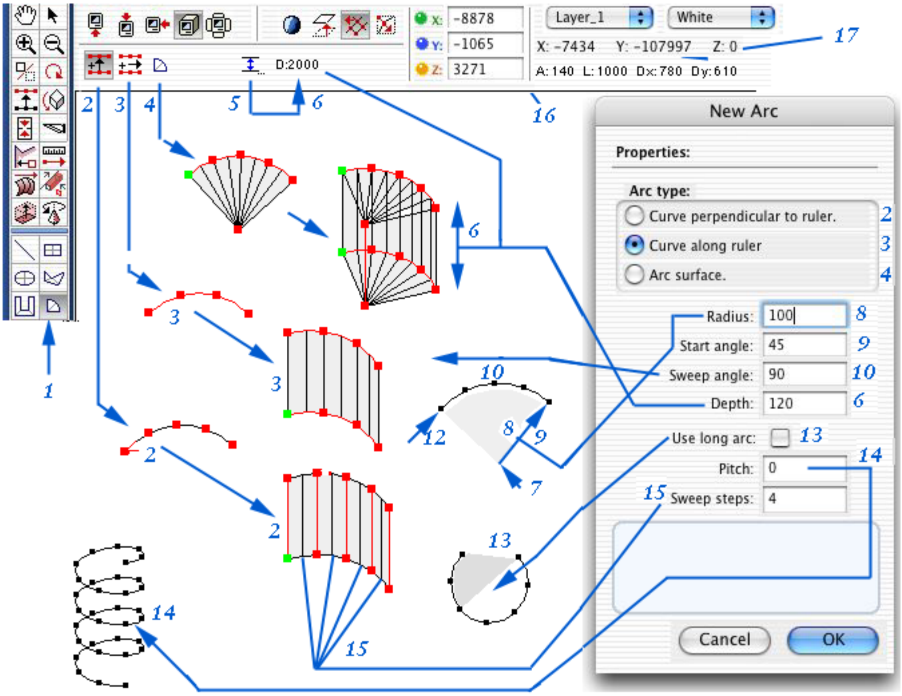
The ***Offset Direction*** *(4)* specifies the location of the double-line relative to the control point polygon.

The ***Plus and Minus*** *(5)* are used for adding or deleting control points. ***The Up and Down*** *(6)* buttons control wherein the ***control point list*** objects are ***being added or deleted.***

The ***Points used*** and ***Current point*** *(7)* lead texts specify where you are in the control point list.

Arc tool

The *Arc (1)* tool in the Blue section of the ***Tool*** palette is used for drawing basic ***2D-arcs*** and ***arc surfaces***. It can also be used for generation ***arc based objects*** if the ***depth*** setting is set to a value ***higher than zero***. Below is a picture showing the steps.



An Arc is drawn using the *Arc (1)* tool in the ***Tool*** palette The Mode bar provides three ***drawing options***, ***Curves*** *(2)* (a series of ruler starting points), ***Control Rulers*** *(3)* and ***Surface*** *(4)*, wherein the surface object consisting of two rulers.

The ***Depth*** *(5)* button displays a ***depth dialog*** where the depth is set. The ***Depth*** *(6)* controls if you want the ***Arc*** to be ***extruded*** or not. A ***depth*** of ***zero*** gives ***no extrusion*** whereas a ***depth higher than zero*** adds an ***extrusion depth*** to the arc. The ***current depth setting*** *(6)* is displayed in the ***Mode*** bar. The ***location*** of the arc in the direction ***away /towards*** you is controlled with the ***three coordinate edit fields*** *(16)* above the drawing area. If you are, for example, in the ***Top*** view while drawing the arc, and with no 3D snapping activated, the ***arc*** ends up at the current ***Y coordinate field*** *(16)* setting. TouchCAD also displays various additional ***dimensional data*** *(17)* while drawing an arc.

To draw an arc, click on ***the center-point*** *(7)* to start the arc. Draw the ***radius line*** *(8)* and click again. Draw the ***arc*** *(10)* and ***click*** *(12)* to finish the drawing procedure.

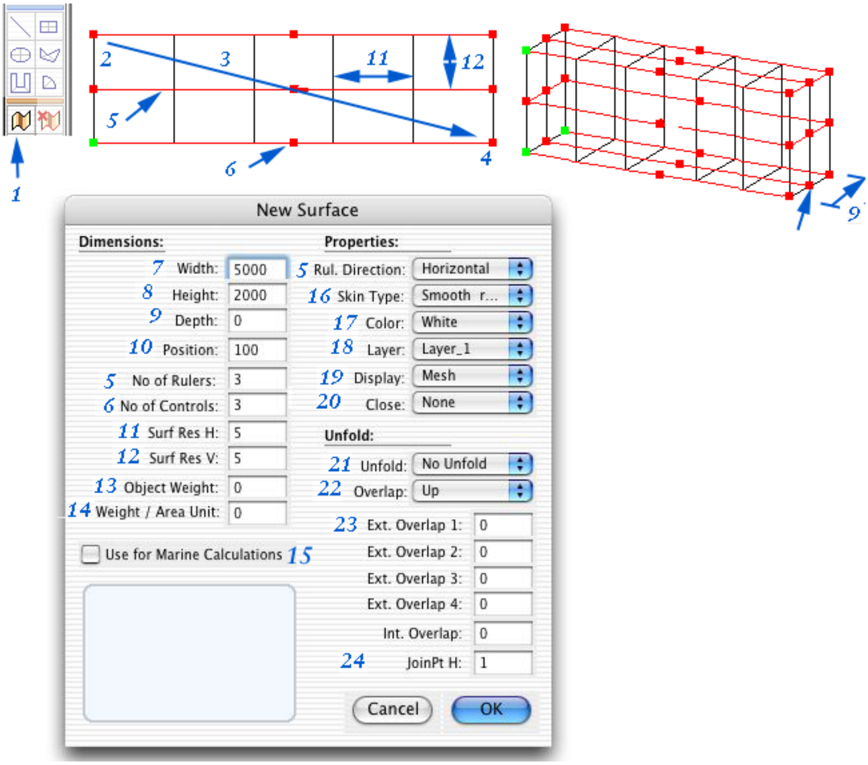
The ***New Arc dialog*** occurs where you are able to define the arc in detail. You can change the ***arc type (2-4)***, ***radius (8), start angle (9), sweep angle (10), depth (6),*** if you want to use the ***long arc (13)*** being more than 180 degrees, the ***pitch (14)*** setting and if you want to add more ***control points (15)*** than the suggested default setting.

Note that ***surface circles / ellipses*** being drawn ***with a depth build a group consisting of several surfaces. Curve and ruler*** based ***objects*** consist of a ***single surface***, and do not have a bottom or top surface. To ***edit*** individual control points in the ***group***, choose the ***Ignore Groups*** mode command in the ***Edit*** menu or use the ***Ungroup*** command in the ***Edit*** menu.

Double-clicking on the Arc tool

You can also draw an arc numerically by ***double clicking*** on the ***Arc*** tool, or by selecting the ***Arc command*** in the ***Tool*** menu as described above.

Surface tool



The ***Surface (1)*** tool in the yellow section of the ***Tool*** palette is used for drawing basic ***3D-surfaces*** and surfaces with a thickness. Such surfaces are usually used as a base for more complex shapes and the dialog window therefore displays a wide range of options to avoid work later in the design process.

The drawing process starts by selecting the ***Surface (1)*** tool in the tool palette. ***Click (2)*** on the ***starting point*** and drag a ***diagonal (3)*** line to the opposite corner of the rectangle and ***click (4)*** again to ***finish the drawing sequence***.

The New Surface dialog occurs. The ***Width (7)*** and ***Height (8)*** specifies the basic dimensions of the rectangle. The ***Depth (9)*** can be set to ***zero*** to build a single surface with ***no thickness*** or a ***six-surface object group*** if the ***depth*** is ***bigger than zero***. The ***Position (10)*** field specifies where to place object in the direction ***away / towards*** you. The ***Coordinate fields*** above the drawing area in the main drawing window define the ***default position*** setting.

The ***No of Rulers** (5)* and ***Controls** (6)* specify the number of ***control rulers*** and ***points*** to be used in the surface. The ***Surf Res H** (11)* & ***V** (12)* fields specifies the ***mesh resolution*** of the surface. Note that the ***Surface Type** (16)* popup may override these settings in some cases.

The two ***weight** (13 / 14)* fields specify the ***weight*** property of the surface. Note that only one of these fields can have a value bigger than zero. The field having a value bigger than zero indicates if the weight per area unit or object weight method is to be used when performing weight calculations.

The ***Marine Calculation** (15)* check box specifies if the surface is to be ***used for marine calculations***. The marine calculation methods are quite mathematically intensive and it is therefore advisable to have this option checked only on surfaces that are actually used for such a purpose.

The ***Ruler Direction** (5)* popup allows you to place the control rulers either ***horizontally or vertically*** relative to the view used when drawing the surface.

The ***Surface Type** (16)* popup specifies the ***smoothing*** settings, and the options are Straight along the Rulers / Straight perpendicular to the rulers, Smooth along the Rulers / Straight perpendicular to the rulers, Straight along the Rulers / Smooth perpendicular to the rulers, and Smooth along the Rulers / Smooth perpendicular to the rulers.

The ***Color** (17) **and Layer** (18)* pop-ups place the new object in the right ***layer*** and gives it a ***color***.

The ***Display mode** (19)* pop-up specifies the ***display mode*** for the object. The options are contours, mesh, cross sections, curves along or perpendicular to the rulers.

The ***Close** (20)* pop-up allows ***closing*** of the surface along the control ruler, perpendicular to the ruler and in both directions. You will find a more detailed description in the ***Object Info / Properties tab*** section of the manual.

The ***Unfold** (21)* pop-up defines if the surface is to be used for ***unfolding*** or not, and how it should behave when being unfolded. The options are no unfolding, joined horizontal strips, horizontal strips, joined vertical strips, and vertical strips. You will find a detailed description in the ***Object Info / Unfold tab*** section of the manual.

The Overlap** (22)* pop-up specifies the direction of internal overlaps counted from the green starting point of the main panel. The options are Up (away from the green starting point), down (towards the green starting point), and overlaps on both sides. You will find a detailed description in the ***Object Info / Unfold tab section of the manual.

The ***Overlap** (23)* fields specify the overlaps individually for each of the four ***external edge sides*** of the main panel and the ***internal overlap*** in between ***the sub-panels***. The ***external overlaps*** are normally inserted ***clockwise*** around the main panel starting from the green staring point of the main panel. You will find a more detailed description in the ***Object Info / Unfold tab*** section of the manual.

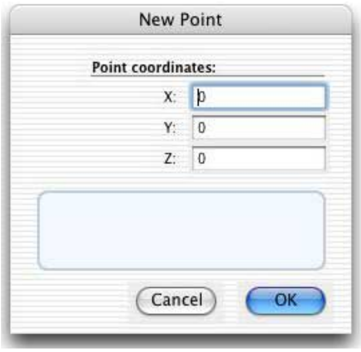
The ***Join At (24)*** field specifies the joining point used when displaying joined unfolded surfaces. The default setting is ***zero*** and indicates that ***the joint is located at the green starting point. Higher values*** move the joining points to ***the respective mesh index point*** (e.g. number five places the joining point at the fifth mesh point). You will find a detailed description in the ***Object Info / Unfold tab*** section of the manual.

Point tool

The ***Point*** tool draws points. ***Points*** can be used for several tasks, for example as a ***reference point***, as a starting point that can be ***extended into a ruler***, ***curve or surface***, or in combination with other points, be ***joined into a curve*** using the ***Combine into Curve*** command in the ***Tool menu***. Points can also be imported using the ***Import -> Points…*** command in the ***File*** menu.

Select the ***Point*** tool in the ***Tool*** palette, and then click in the drawing area to draw a point. The location can be adjusted using the ***Coordinate*** fields above the drawing area, and / or using the ***Arrow*** keys.

Below is a picture showing the ***New Point dialog*** occurring when you ***double-click*** on the ***Point tool***.



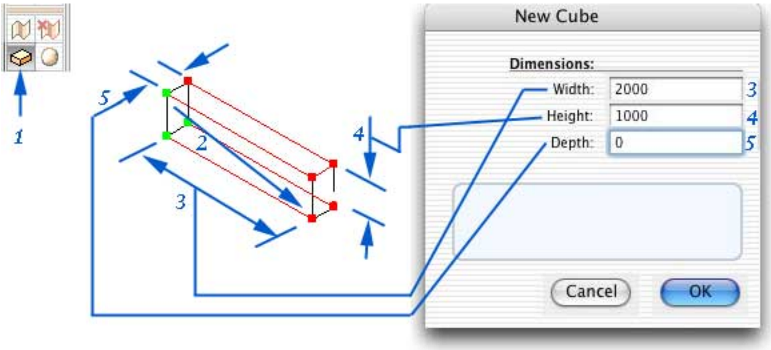
A point can be ***extended into a control ruler*** by clicking on it with the ***Selection Arrow*** tool (***Mesh*** mode) ***while pressing*** the Alt / Opt ***key***. This adds a new control point to the point, and therefore converts it into a control ruler. Adding ***a new ruler*** to the point is done by ***pressing*** the Alt / Opt + ***Shift keys***. This means that a point is essentially an object with only one control ruler that has a single control point, and that is why you can handle it like any other TouchCAD object.

Double-clicking on the Point tool

You can also draw a point numerically by ***double clicking*** on the ***Point*** tool, or by selecting the ***Point command*** in the ***Tool*** menu as described above.

Cube tool

The cube tool draws cubes. Below is a picture showing the steps.



Select the *Cube (1)* tool in the yellow section of the Tool palette. Draw the cube *(2)* in the drawing area. The ***New Cube*** dialog occurs where you specify ***the width** (3)**,*** ***height** (4)* ***and depth*** *(5)* settings. Click OK and the cube occurs.

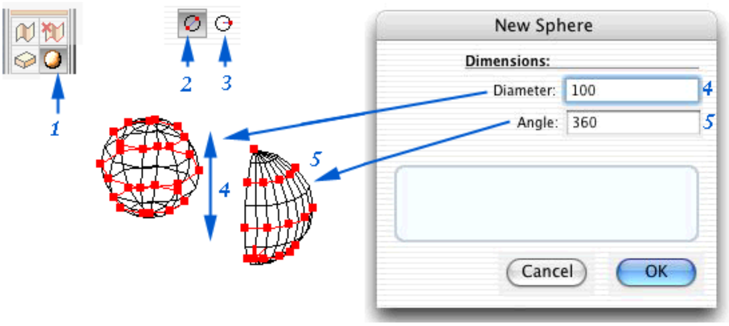
Note that ***cubes*** form ***a group consisting of many surfaces***. To ***edit*** individual control points in the ***group***, choose the ***Ignore Groups*** mode command in the ***Edit*** menu or use the ***Ungroup*** command in the ***Edit*** menu.

Double-clicking on the Cube tool

You can also draw an cube numerically by ***double clicking*** on the ***Cube*** tool, or by selecting the ***Cube command*** in the ***Tool*** menu as described above.

Sphere tool

The *Sphere* tool creates spherical objects.

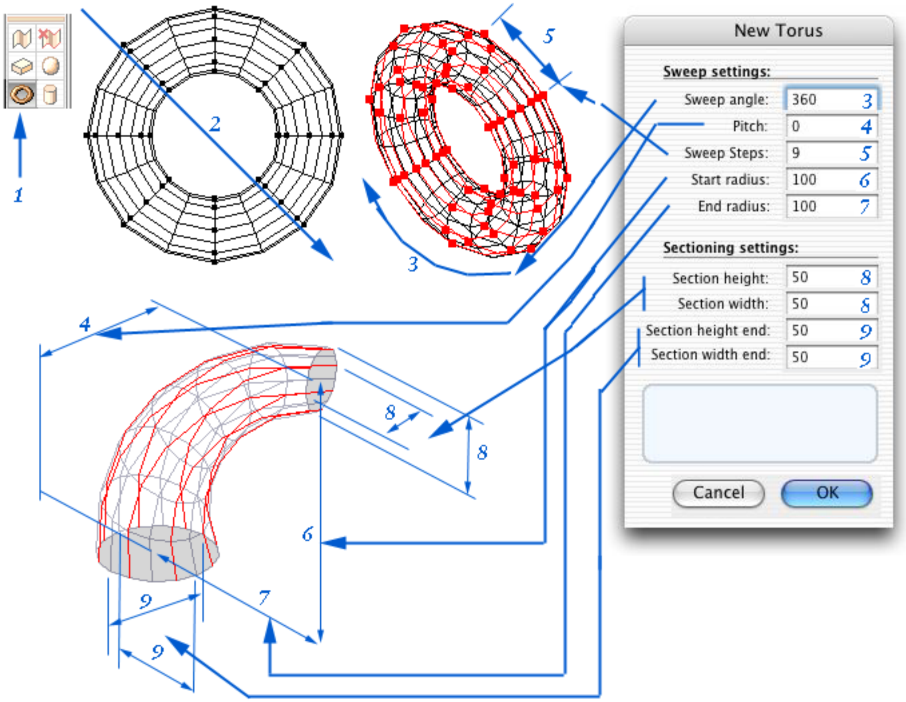


Select the ***Sphere*** *(1)* tool in the Tool palette. Select the ***Diameter*** *(2)* or ***Radius*** *(3)* method in the mode bar. Draw the shape in the drawing area. The new ***sphere*** *(4)* occurs.

An alternative method is to ***double-click*** on the ***Sphere*** *(1)* tool in the ***Tool*** palette. The ***New Sphere*** dialog is shown and here you can enter the ***diameter*** *(4)* and ***sweep angle*** *(5)*.

Torus tool

The *Torus (1)* tool generates ***torus*** based objects such ***as elbow tubes, spirals, coil springs***, etc. Below is a picture showing the steps.



Select the *Torus (1)* tool in the yellow section of the tool palette. ***Click*** in the drawing area, ***drag*** and then ***click again*** to draw the ***basic shape*** *(2)*. The ***New Torus dialog*** is displayed, wherein you have the following options.

The ***Sweep Angle*** *(3)* field controls sweep angle. The uppermost *(2-3)* examples are 360 degrees whereas the lowermost is 90 degrees. Angles higher than 360 degrees can be used though it would mostly make sense in combination with a ***pitch*** *(4)*.

The ***Pitch*** *(4)* field adds a ***pitch***, that is, a ***spiral shape*** in the ***direction away / towards*** you.

The ***Sweep Steps*** *(5)* field controls how many ***control points*** used in the object. The default setting is normally adequate and the only ***reason for changing*** it is if you have specific needs, such as if the object is to be used ***for generating complex transformations*** or if you need an ***extreme degree of accuracy***.

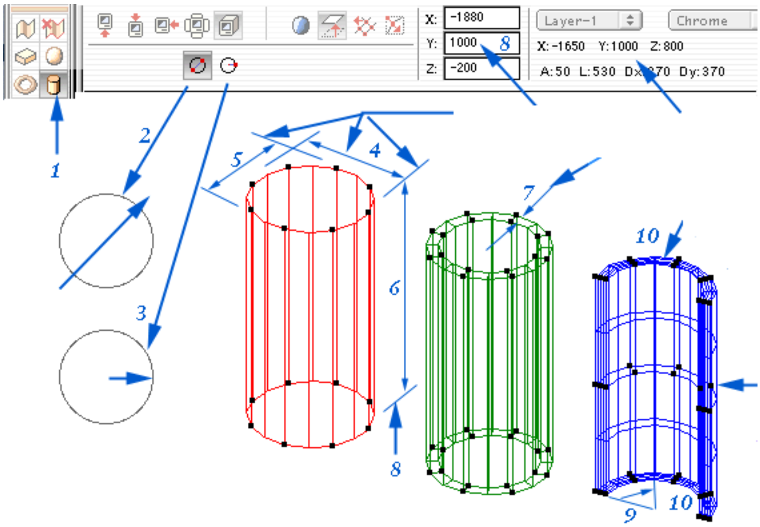
The ***Start Radius*** *(6)* and ***Stop Radius*** *(7)* fields controls ***the sweep radius.*** The default setting is the ***same setting*** for these two fields and generates a sweep having the ***same radius*** across the sweep. If the ***start radius*** is ***smaller*** that the ***stop radius***, the ***sweep*** forms a ***smooth transformation outward*** and away from the center point and visa versa.

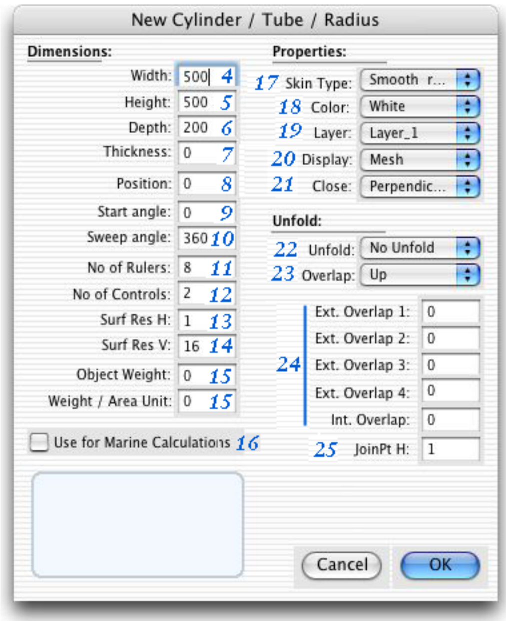
The ***Start and End Width and Height*** *(8-9)* fields allow ***internal transformations*** of the basic ***cross sections*** along the sweep. You can for example have a 100 x 100-mm circular cross section at the start, and that smoothly transforms into a 200 x 300-mm elliptical cross section at the end of the sweep.

Note that the ***Torus*** shape is a normal TouchCAD ***surface*** and it therefore allows you to change the shape both by ***moving the control points*** as well as the settings in the ***Object Info*** palette. You can for example change the number of steps used along the sweep path to make it easier to produce for example a sheet metal based elbow tube.

Tube / Cylinder / 3D Arc tool

The ***Tube / Cylinder / 3D Arc*** tool generates various ***cylindrical*** shapes. Simple cylinders as well as tubes and sections of tubes with a wall thickness can be generated. Below is a picture showing the steps and options.





The ***Tube / Cylinder / 3D Arc (1)*** tool found in the yellow section of the ***Tool*** palette is used for drawing basic 3D-cylinders, tubes etc and surfaces with a thickness. Such surfaces are usually used as a base for more complex shapes and the dialog window therefore displays a wide range of options to avoid work later in the design process.

The drawing process starts by selecting the ***Tube / Cylinder / 3D Arc (1)*** tool in the tool palette. The ***Diagonal (2)*** and ***Radius (3) methods*** are optional methods in the ***Mode*** bar. ***Click*** on the starting ***point*** and drag a line to the opposite corner of the circle and ***click*** again to ***finish the drawing sequence***.

The ***New Cylinder Tube Radius*** dialog occurs. The ***Width (4)*** and ***Height (5)*** specifies the ***basic dimensions*** of the cylinder. The ***Depth (6)*** field defines the height of the cylinder in the direction away / towards you. The ***Wall thickness (7)*** field can be set to ***zero*** to build a ***single surface*** with no wall thickness, or ***a surface object group*** if the ***thickness*** is ***bigger than zero***. The ***Start Position (8)*** field specifies ***where*** to place object in the ***direction away / towards*** you. The ***Coordinate*** fields above the drawing area in the main drawing window defines the default position setting.

The ***Start angle (9)*** defines the ***angle of the circle /***arc starting angle and the ***Sweep angle (10)*** defines how many ***degrees the sweep arc*** is.

The ***No of Rulers (11)*** and ***Controls (12)*** specify the number of ***control rulers*** and ***points*** to be used in the object. It is normally best to leave these fields with their respective default settings, unless you want to use the shape as a base for a ***curved surface*** or if you wish to use ***curved edge surfaces***.

The ***Surf Res H (13)*** & ***V (14)*** fields specify the ***mesh resolution*** of the surface. Note that ***Surface Type (17)*** popup may override these settings in some cases.

The two ***weight (15)*** fields specify the ***weight property*** of the surface. Note that only one of these fields can have a value bigger than zero. The field having a value bigger than zero indicates if the weight per area unit or object weight method is to be used for this particular method when performing weight calculations.

The ***Marine Calculation (16)*** check box specifies if the surface is to be used for marine calculations. The marine calculation methods are quite math intense and it is therefore advisable to have this option only checked on surfaces that are actually used for such a purpose.

The ***Surface Type (17)*** popup specifies the smoothing settings, and the options are Straight along the Rulers / Straight perpendicular to the rulers, Smooth along the Rulers / Straight perpendicular to the rulers, Straight along the Rulers / Smooth perpendicular to the rulers, and Smooth along the Rulers / Smooth perpendicular to the rulers.

The ***Color (18) and Layer (19)*** pop-ups place the new object in the right layer and gives it a color.

The ***Display mode (20)*** pop-up specifies the display mode for the object. The options are contours, mesh, cross sections, curves along or perpendicular to the rulers. For a more detailed description see the ***Object Info palette / Properties tab*** section of the manual.

The ***Close (21)*** pop-up allows closing of the surface along the control ruler, perpendicular to the ruler and in both directions. For a more detailed description see the ***Object Info palette / Properties tab*** section of the manual.

The ***Unfold (22)*** pop-up defines if used for unfolding or not, and how it should behave when being unfolded. The options are no unfolding, joined horizontal strips, horizontal strips, joined vertical strips, and vertical strips. You will find a more detailed description in the ***Object Info palette / Unfold tab*** section of the manual.

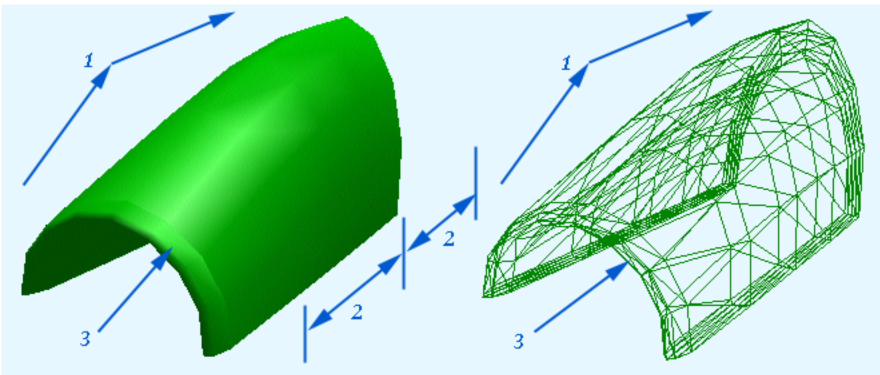
The Overlap (23) pop-up specifies the direction of internal overlaps counted from the green starting point of the main panel. The options are Up (away from the green starting point), down (towards the green starting point), and overlaps on both sides. You will find a more detailed description in the ***Object Info palette / Unfold tab*** section of the manual.

The ***Overlap (24)*** fields specify the overlaps individually for each of the four external main panel sides and the internal overlap in between the sub-panels. The external overlaps are normally oriented clockwise around the main panel starting from the green starting point of the main panel. You will find a more detailed description in the ***Object Info palette / Unfold tab*** section of the manual.

The ***Join At (25)*** field specifies the joining point used when displaying joined unfolded surfaces. The default setting is zero and indicates that the joint is located at the green starting point. Higher values move the joining points to the respective mesh index point (e.g. number five, places the joining point at the fifth mesh point). For a more detailed description se the ***Object Info palette / Unfold tab*** section of the manual.

Double-clicking on the Tube / Cylinder / 3D Arc tool

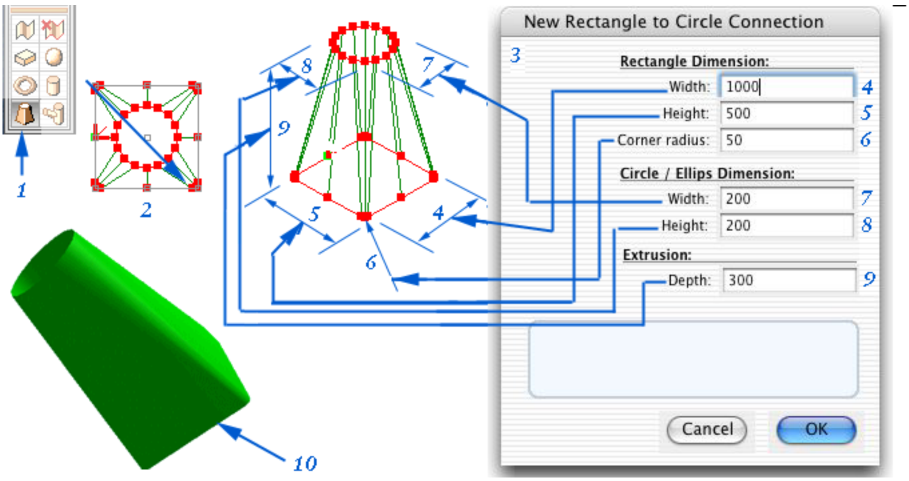
You can also draw an cylinder numerically by ***double clicking*** on the ***Tube / Cylinder / 3D Arc*** tool, or by selecting the ***Cylinder command*** in the ***Tool*** menu as described above.



Above is an example showing an object founded on a basic 180-degree tube object used as a starting object for a more complex shape. The ***height in the back*** was increased (***1***) to generate a convex shape. This was achieved by adding an ***extra control point*** in the middle of the surface (**2**). The shape also has a ***curved front edge (3).***

Rectangle to Circle tool

The ***Rectangle to Circle Connection*** tool is used for generating smooth transformations between a rectangular and a circular or elliptical cross section. It also allows you to add a small radius in the rectangular part, for example to simulate the bending radius generated when bending sheet metal objects. The basic shape can also be transformed to handle eccentric transformations or where the cross sections are not parallel to one another. Below is a picture showing the steps.



Select the ***Rectangle to Circle*** *(1)* in the yellow section of the ***Tool*** palette. Click and drag a diagonal line and then click again to draw the ***basic object*** *(2)* in the drawing area. The ***Rectangle to Circle Connection dialog*** *(3)* window occurs.

The ***Rectangular Width*** *(4)*, ***Height*** *(5)* and ***Corner Radius*** *(6)* fields specify the properties on the rectangular side of the object.

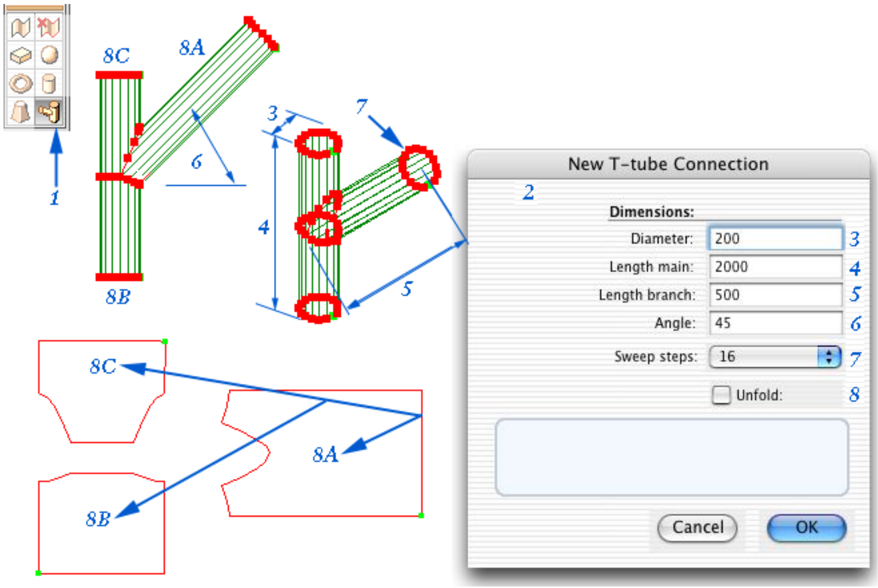
The ***Circular Width*** *(7)* and ***Height*** *(8)* fields controls ***the circular side***.

The ***Extrusion Depth*** *(9)* field specifies the ***length*** of the extrusion.

Note that it is often practical to add a small ***radius*** *(6)* in the rectangular end of such a shape, especially if the object is to be used for producing sheet metal objects. Sheet metal can normally not be folded into an absolutely sharp edge in a corner, so in most cases it would look more like the rendered picture *(10)* when produced as a real physical model.

T-tube tool

The *T-tube* tool draws T-shaped tube connections. Below is a picture showing the steps.



Select the *T-tube (1)* tool in the yellow section of the *Tool* palette. Click on the drawing area. The *T-tube Connection (2)* dialog occurs.

The *Diameter (3)* field specifies the ***diameter*** of both the ***main parts*** and the ***branch***. The *Length Main (4)* field specifies the total ***length of the main tube***. The *Length Branch (5)* field defines the ***length of the branch*** component. The *Angle (6)* field defines the ***angle between the main tube and the branch***. The ***maximum angle*** is ***90 degrees*** and the minimum is defined by what fits in along the main tube without extending upwards beyond the upper edge of the main tube. If such an event occurs, TouchCAD displays a warning message and then calculates the lowest possible angle given by the current settings. You can however decrease the angle further by adding more length to the main tube. The *Section Steps (7)* enable you to choose between a number of ***resolution settings***. A higher resolution generates smoother looking unfolded curves but also requires more power to calculate it. The *Unfold (8)* checkbox specifies that the shape is to be used for unfolding purposes and enables you to see the unfolded patterns directly after having selected the *Unfold view (8 A-C)*.

Note that *T-tubes* form ***a group consisting of several surfaces***. To ***edit*** individual control points in the ***group***, choose the *Ignore Groups* command in the *Edit* menu or use the *Ungroup* command in the *Edit* menu.

T-tubes can also be used as a foundation for generating other similar objects such as T-tubes with variable cross section transformations, etc.

File menu

The file menu contains standard features found in many other programs, for example Open, Save, Print and Quit.

New

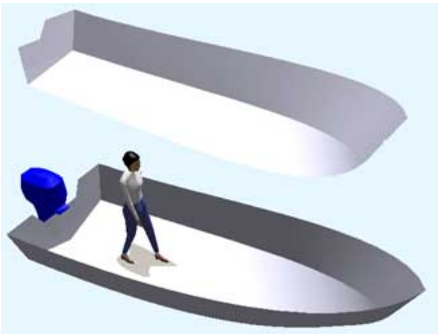
The *New...* command opens a new empty document. If an unsaved document is active, TouchCAD displays a warning dialog, wherein you have the option to either save the current document or go directly to a new and empty document without saving first. The ***quick key*** for the ***New*** command is ***[Command + N (M) / Control + N (W)].***

Open

The ***Open...*** command opens generic TouchCAD documents. If an unsaved document is active, TouchCAD displays a warning dialog, wherein you have the option to either save the current document or go directly to a new and empty document without saving first. The quick key for the ***Open*** command is ***[Command + O (M) / Control + O (W)].***

Load Into...

The *Load Into...* command ***opens*** generic TouchCAD documents ***and inserts*** the content thereof into the currently active document. The purpose of this command is primarily to ***insert previously drawn object elements***. It enables you to create a library of predefined objects such as people, trees, cars, furniture, engines, etc, and use them as reference objects in you projects. An example is shown below.



Importing an outboard motor and a person immediately creates a sense of proportion to the rendered image.

Consider adding a new layer for such objects before importing them. In this way you can easily turn them off and on while editing the model.

Save

The *Save* command ***saves*** the active document. If the document has not been saved before, the *Save As* dialog occurs, wherein you are prompted to enter a document name. The ***quick key*** for the *Save* command is *[Command + S (M) / Control + S (W)].*

Save As...

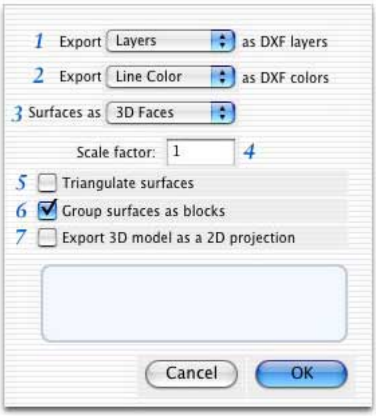
The *Save As...* command saves the active document under a new name. The *Save As dialog* occurs, wherein you are prompted to enter a document name.

Revert to saved

The *Revert to Saved* command takes you back to the last saved version of the model.

Export as -> DXF...

The *Export as -> DXF...* command ***exports*** the active model as a *DXF file*. A DXF file is a standard file format defined by AutoDesk, producers of AutoCAD, and is used for exchanging files between various CAD and 3D-modeling programs on both the Windows and Mac platforms.



The ***DXF export dialog*** contains the following items.

The ***Export DXF layers*** *(1)* popup allows you export TouchCAD layers, colors or objects as DXF layers. When using the ***TouchCAD layers*** you keep the layer structure from TouchCAD. Exporting ***TouchCAD colors as DXF layers*** is primarily useful when exporting 3D models to ***rendering programs*** where layers are used for defining the color identity of the objects. TouchCAD supports 256 DXF colors when exporting and ***uses the DXF color*** being closest to the color definition in TouchCAD. Exporting ***objects in separate layers*** can be useful when exporting to some rendering programs and when sending files to some plotter driver programs that do not support groups.

The *Export Line Color (2)* popup allows you to use either the ***line or fill colors*** for export to DXF. The reason for this option is that DXF does not separate the line and fill colors, so you simply need to choose either of these two options.

The ***Surfaces As (3)*** popup allows you to choose between exporting surfaces as the DXF data types ***3D Faces*** or ***Polygon Mesh***. Choose the one that works best in the receiving program. TouchCAD can read both these data types but Polygon Meshes usually fits better into the internal data handling as it builds larger units rather than individual surface elements.

The ***Scale factor (4)*** allows you to ***scale the model*** when exporting. It is primarily useful when exporting for example a model drawn in mm to a document based on inches. It can also be useful when exporting to drawing programs, which do not support full scale drawing, a drawback with many illustration programs.

The ***Triangulate (5)*** option is primarily used when the model is to be used for ***rendering purposes***. Most rendering programs base their models on triangles and it can therefore be practical to convert to triangles in such cases.

The ***Group Surfaces as Blocks (6)*** option generates a ***DXF datatype*** called ***Anonymous Blocks*** (the equivalent of ***groups*** in most other programs) and makes it easier to handle the objects in such other programs. Note though that not all programs support this feature, so don't use it if it causes problems.

The ***Export 3D models as a 2D projection (7)*** option exports ***the 3D views as seen on the screen*** and not as a 3D model. This option is primarily used when exporting to strictly two-dimensional programs.

Export as -> EPSF

The ***Export as EPSF*** command is used for exporting 2D drawings to ***Adobe Illustrator***, and other programs compatible with this file format, such ***as MacroMedia Freehand, Canvas, CorelDraw and PhotoShop***. The exported files are fully ***editable*** in the receiving program if you use the ***Open*** method. Both ***3D views*** and ***unfolded patterns*** can be exported. 3D-models are exported as a ***2D-projection*** as seen on the screen and the reason for this is of course that the receiving programs are all essentially 2D-illustration programs.

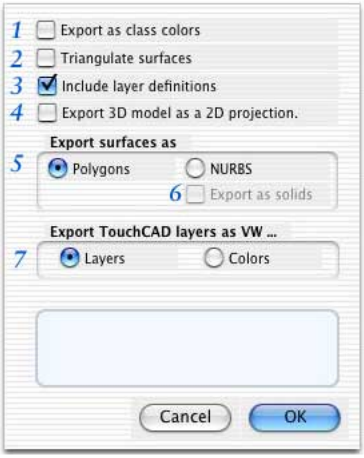
It is important to know that TouchCAD always works using full-scale coordinates and uses the Drawing Scale to reduce the size of the model to fit into a given ***sheet*** space. This is not the case with most illustration programs, and such programs often have a limited drawing area. TouchCAD therefore ***exports*** to the ***EPSF format*** using the ***scale factor*** stated in the ***Drawing Scale dialog***.

Export as -> 3DMF

3DMF (QuickDraw3D) is a file format defined by Apple Computer and is used for ***displaying rendered 3D models***. It allows fast real-time rotation of models. The QuickDraw3D format is available for MacOS Classic and Windows but is not supported in the new Mac OSX unless you install the Quesa plug-in in your system. TouchCAD uses QuickDraw3D models in the internal rendering engine.

Export as -> VectorWorks

The ***VectorWorks export*** command exports files to ***Nemetschek's VectorWorks*** and ***MiniCAD*** (older name for VectorWorks) in the ***VectorScript file format*** (called ***MiniCad Text*** in MiniCad 7 and older versions). Drawings can be exported as ***3D models***, as ***projected 2D*** version of the 3D models, or as ***unfolded patterns***. Below is a picture showing the ***Export dialog*** options.



The ***Export as Class colors (1) checkbox*** generates a ***specific Class name and color*** for each TouchCAD color used in the model. If the checkbox is ***off***, TouchCAD simply ***exports*** the ***colors with no name***.

The ***Triangulate (2) checkbox*** allows you to sub-divide all surface objects into ***triangles***. Such a conversion is especially useful if you intend to use the generic VectorWorks rendering engine plug-in called ***RenderWorks***, which does not handle twisted quads (four-sided polygons) well.

The ***Include layer definitions (3) checkbox*** allows you to ***export*** the ***layers or colors*** used in TouchCAD as ***VectorWorks layers or colors***. Deselect this option if you just want to import the entire TouchCAD model into the currently active VectorWorks layer or class.

The ***Export 3D model as a 2D-projection (4) checkbox*** allows you to export a 2D-representation of a 3D-model.

The ***Polygon / NURBS (5)*** options defines the data types used when exporting. The Polygon option simply exports all objects as polygons, whereas the NURBS option exports curves and surfaces as NURBS objects. The later is better if the purpose is to continue to work with smooth looking objects, when there is a need to perform solid modeling calculations in VectorWorks or when there is a need to re-export objects using the VectorWorks IGES export features. When the NURBS option is activated, the ***Export as Solids (6)*** checkbox becomes selectable, which enables you to export surfaces as solid or just plain NURBS surfaces. The NURBS option only works on VectorWorks 10 or later.

Export as -> VRML

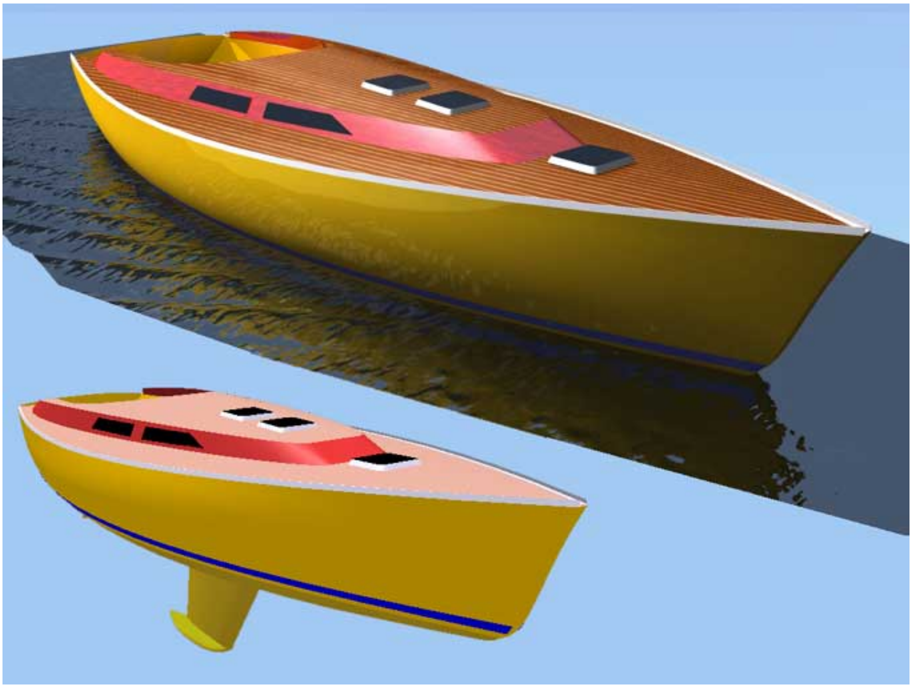
The *Export VRML command* exports 3D models as *VRML files*. This format is primarily used for displaying 3D models in real-time. Many rendering programs support this format.

Export as -> StrataVision

The *Export as StrataVision command* exports files in a simplified *MiniCAD text* format, which can be read by many versions of the Strata line of rendering programs. Strata also reads DXF, VRML and 3DMF as an intermediate format for exporting.

Export as -> Artlantis 3-4 / 4.5

The *Export as Artlantis commands* are used for exporting 3D models to various versions of the *rendering program Artlantis*. The *3-4 option* exports as *.opt / .db* files whereas the *4.5* have a new generic *format called .atl*. TouchCAD *exports* the *colors* used as *Artlantis color objects* and includes both *fill color* and *transparency* settings. The imported model scene is *predefined* having the *shadows*, *reflectivity* and *transparency* options activated, and with a *default light object* and *camera setting*. This means that the model is fairly well prepared from the start and only needs some basic adjustments to be ready to build a basic photo realistic rendering. Artlantis is available for Mac and Windows. More info and demos can be found at www.artlantis.com. Below is a picture showing the difference in rendering quality between the generic TouchCAD rendering engine and an Artlantis rendering.

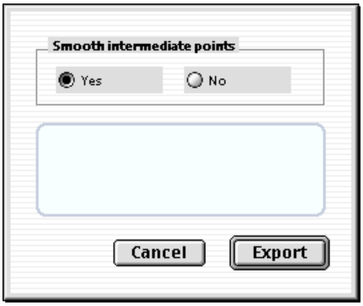


Export as Plotmaker

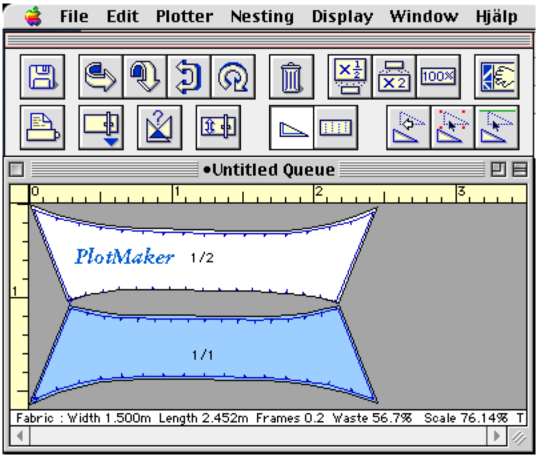
The *Export as Plotmaker* command exports both ***projected 3D views*** and ***unfolded patterns*** to ***the sail-cutting program PlotMaker*** developed by ***Sail Science*** in New Zealand. Sail makers using the *SailMaker* program can use their hardware for producing other fabric-based objects such as canvas elements, sail-roofs, blimps, etc.

When exporting from a 3D view, TouchCAD exports the view as a 2D projection. The export dialog enables exporting as cut or as plot lines. The primary use is to generate graphics and convert it into objects that can be processed in the plotter/cutter.

When being in the Unfold view, TouchCAD supports PlotMaker export of ***all unfolding options*** in the *Object Info* palette and most options in the ***View -> Show in Unfold*** menu. The exceptions are that it does ***not*** export ***coordinates*** and ***line /fill colors***, because it does not make sense to export these features to a strict cutting program.



An export dialog occurs after having selected the ***Export->Plotmaker*** command in the ***File*** menu. The ***Smooth*** option assumes that you let ***Plotmaker do the required smoothing*** between the vertex points. The ***Straight*** option exports the model exactly as ***seen in TouchCAD***. It can be hard to say which of these options that works best in a given situation, but a good strategy may be to use the ***Screen Plot*** command to simulate the cut. If that looks right there is a good chance that the actual cut will work as well.

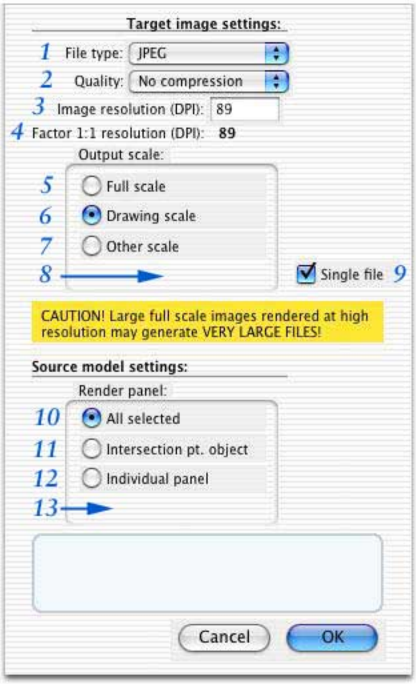


TouchCAD ***exports*** the unfolded model in the ***PlotMaker ASCII format***, which can be imported by simply double-clicking on the file generated, or by selecting ***the Import->PlotMaker ASCII*** command in the ***File*** menu of ***PlotMaker***.

Export as Unfolded images.

Background images can be mapped on the 3D model and be saved as a single or a set of images. To be able to do it you have to prepare a model and image as follows.

Start by importing a ***background image*** and place it behind a 3D model, which must consist of surfaces. Make sure that the surfaces to be used have ***an unfolding property*** (other than “No Unfold” in the Object Info palette -> Unfold Tab -> Unfold pop-up). ***Select the object*** or objects to be used for the image unfolding calculation. ***Select the view*** where you can see the background image and then ***select Unfold Images*** from the ***File*** menu. The following dialog occurs.



The ***File Type*** *(1)* pop-up states the file type to be used.

The ***Quality*** *(2)* pop-up states the degree of compression used.

The ***Image Resolution*** *(3)* edit field states a target resolution for the unfolded image or images. The value stated in the example suggests 89 DPI, which roughly reflects the resolution of the image relative to the scale stated in #5 to #8. In the example using the drawing scale. Click on the ***Full Scale button*** *(5)* and the DPI rate changes provides that drawing scale in not 1:1. The lead text below (4) suggests a scale, which can be used or not. It is fully possible to scale an image upwards though you it must be said that you may loose some in quality as TouchCAD only calculates an image based on the original. Worth noting is that very physically large images are typically printed at much lower resolutions than used in form example brochures, magazines and books, sometimes as low as 8 to 12 DPI. This works because you need to back off quite a lot to actually make sense of say a 30-meter wide image. In other words, use the DPI settings with caution. If you double the rate you typically quadruple the file sizes generated and rendering times. It may not matter much if the file is 200K and you go to 800K but if you start at 75 Megabytes you end up with 300 Megs if you double the setting. At some point there is a risk that you simply run out of both RAM and hard disk space. In other words, use it with ***caution***.

The ***Output Scale*** buttons controls the output scale using the ***Full Scale*** *(5)*, ***Drawing Scale*** *(6)* or ***Custom Scale*** *(7)* options. If the later is selected an ***edit field*** *(8)* is displayed where you enter a ***custom scale***.

The ***Single File*** *(9)* checkbox controls if the unfolded image(s) is to be exported as a single file or as separate images for each of the chosen panels. Note that it may be a good idea to create a special folder for projects where you unfold many panels as 50 or 100 new files in an existing folder may cause a bit of a mess. When images are exported as individual files you start with a basic file name, for example called "MyImage". Each new image is then named MyImage + the panel and sub panel index number, for example "MyImage_1_12.jpg". This makes it easier to identify a sub panel without actually having to open it.

The Render Panel buttons control which panels to be sued for a particular rendering session. ***All Selected*** *(10)* render all panels selected in the drawing window. Intersection ***Point Object*** *(11)* renders the surface that carries the Intersection Point symbol and all it's sub parts. The ***Individual Panel Option*** *(12)* just renders a single sub panel based on its ***object*** and ***sub panel ID*** as stated in the ***edit fields*** *(3)* below the button (not visible in the diagram).

Once the paper work is done it is just a matter of clicking OK and wait for the result.

The final step in the process is to open the file(s) and change the resolution to the target resolution as stated in the export dialog. The files are all exported at 72 DPI by default but if you change it to the target resolution you get images that match the stated scale. In ***PhotoShop*** you use a command called ***Image Size*** found in the ***Image*** menu.

Worth mentioning is that the image unfolding also adds all the drawing elements seen in the Unfold view, such as cut and plot lines, panel numbers, alignment marks, pen and line color settings etc. In other words all settings as defined in the Object Info palette as well as the general unfolding settings as defined in the Show in Unfold commands found in the View menu.

Images that only partially cover the unfolded model are only rendered where they are located whereas uncovered areas use the color settings as seen in the Unfold view. If a given area has a white fill setting it simply generates an image with a white fill in the image.

To save space and keep the image sizes to a minimum it is recommended to edit the unfolded parts a bit in the Unfold view to minimize the area used. TouchCAD simply generates images that found boundary boxes around the unfolded shapes.

Export as HPGL

The Export as HPGL command is used for exporting HPGL file. The file format was originally used for controlling pen plotters but is nowadays mostly used for computer controlled cutting tables and vinyl cutters.

Export as STL

The *Export as STL* command exports *3D-models* as STL files. The file format is primarily used for *quick prototyping and 3D-milling*.

Export as Image File.

The *Export as Image File* command exports pixel based screen dumps as image files. The export dialog enables you to export as JPEG, TIFF, TARGA, PICT, BMP, and PhotoShop, using the *File Type (1)* pop-up. By default it generates a screen dump, which is what the *Width and Height (2)* edit fields states measured in pixels. Changing the numerical values can scale these settings upwards and downwards proportionally.

Export as Points

All objects in the drawing can be exported as *text based TAB* formatted file.

Importing files.

Files can be imported into TouchCAD in DXF, VectorWorks (VectorScript), Points, STL, SailMaker ASC 3D, Wavefront, QuickDraw3D, VRML formats. TouchCAD can also import in-scale background images used as a modeling aid and for generating unfolded images.

Import as ->

The *Import As* sub-menu contains import filters for various file formats such as DXF, VectorWorks (VectorScript / MiniCAD text), Points, STL, Sails Science ASC 3D, Wavefront, VRML, QuickDraw3D (text) and as in scale Background Images..

Import as -> DXF

The ***Import as -> DXF***... command imports ***DXF file***s. A DXF file is a standard file format defined by AutoDesk, producers of AutoCAD, and is used for exchanging files between ***various CAD and 3D-modeling*** programs on both the Windows and Mac platforms.

TouchCAD supports importing of the following DXF ***data typ***es: Lines, circles, arcs, polylines, LW polylines, Polygon Meshes, Polyface Meshes, 3D faces, points, layers, 256 colors and anonymous blocks (but not blocks).

The ***DXF import dialog*** allows you to change ***flip the Y and Z directions*** *(1)*, which is used when importing from programs having a flipped coordinate system.

A ***scale factor*** *(2)* can be applied to the import model, which is used when the file is exported in another drawing unit than used in the current drawing. A model drawn in meters would for example need a scale factor of 1000 if the drawing units is set to millimeters. The scale factor has to be set manually because the DXF format does not specify the drawing units.

Import as -> VectorWorks

The ***VectorWorks import*** command imports files from ***Nemetschek's VectorWorks and MiniCAD*** (the older name for VectorWorks) in the ***VectorScript*** file format (called ***MiniCad Text*** in MiniCad 7 and older versions).

TouchCAD supports the importing of the following VectorWorks ***data types***: Lines, circles, ovals, arcs, rectangles, 2D polygons, 3D polygons, 2D & 3D locus points, layers and fill / line colors.

Import as -> Points

The ***Points*** import format is a ***simple text-based TAB based X, Y, Z, coordinate list***. The syntax is a s follows:

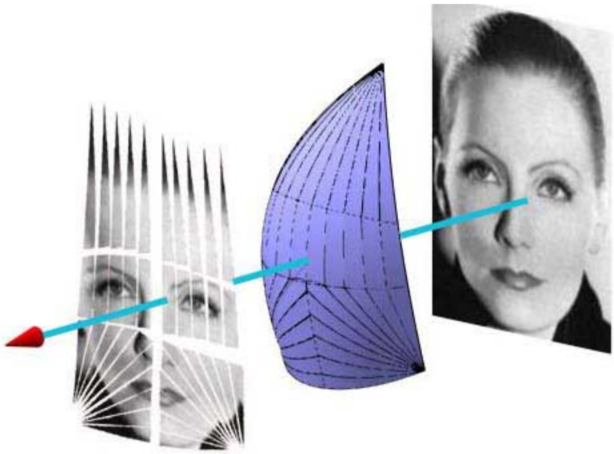
123.123 ***(TAB)*** 456.456 ***(TAB)*** 789.789 ***(Carriage Return)***

789.789 ***(TAB)*** 456.456 ***(TAB)*** 123.123 ***(Carriage Return)***

Etc.

Such a file can be generated using an ordinary word processor, spreadsheet or database program and the typical use is to import and make use of measurement points to create a model.

The manual ***section*** concerning the ***Combine Into Curve*** command contains an example on how to process and refine points into usable surfaces.



Import as -> Sailmaker ASC 3D

The import Sailmaker ASC 3D format is used for importing 3D sails from Sails Science Sailmaker. The main use if to allow further processing of sails within TouchCAD, for example to apply images and unfold then to produce sails with printed images such as shown above. Sails imported maintain their original panel layout and they are pre-set to be unfolded at their original location used in Sails Science Plotmaker.

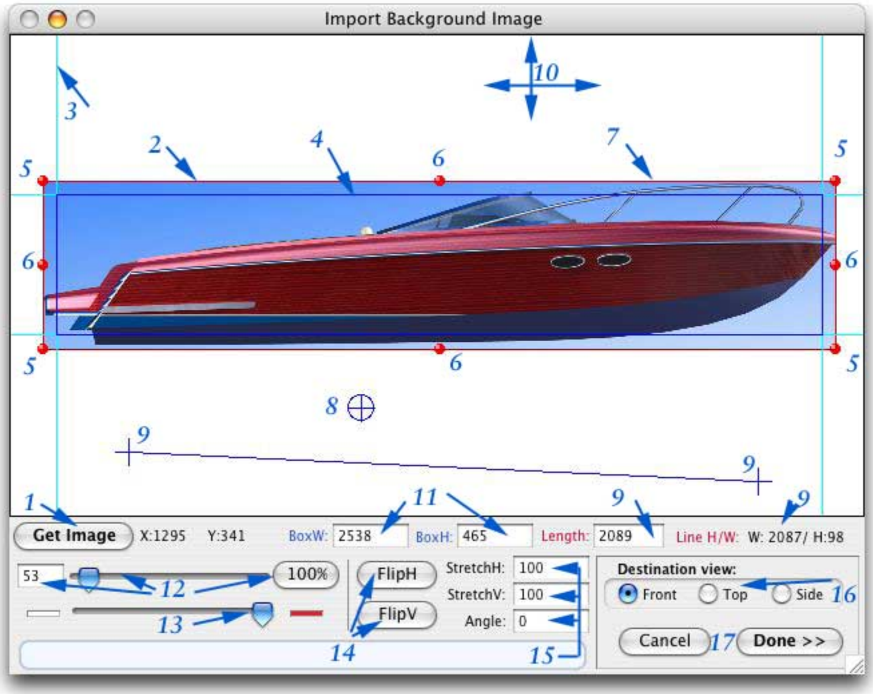
Import as -> STL / Wavefront / QuickDraw3D / VRML

It is possible to *import STL / Wavefront / QuickDraw3D / VRML files* to be used as reference objects. In most cases though, such models tend to contain lots of triangles and are therefore not particularly well suited for use in TouchCAD. The reason for this is that TouchCAD models are based on mathematically defined surfaces that forms large blocks of surface elements, which is a very different approach. It can however be useful to use such models when designing for example inflatable objects as there are quite a lot of predefines models available on the Internet. In such cases you usually need to re-skin and simplify the model to reach a truly usable compromise between shape and making it reasonably easy to build.

Import -> Background Image

The import background image feature enable importing of high resolution background images that can be used both as a modeling aid as well as for generated unfolded images. Background images are imported in exact scale relative to the 3D model and follow the 3D model when panning and zooming. It is also possible to import separate background images for the Front, Top and Side views to generate corresponding 2D view images that can be compiled into a real 3D shape. Background images are by nature connected to given standard view and is only shown when the relevant vise is selected.

When background images are to be used for modeling and image unfolding, it is usually a good idea to check the dimensions of the 3D model to be used and to make a note of it for use later on in the process. The diagram blow shows the import dialog displayed after selecting the Import Background image command.

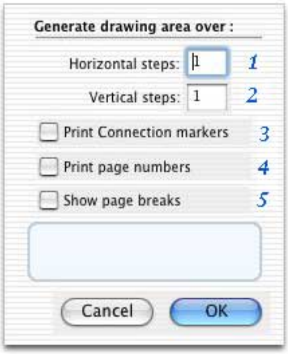


The first step is to select the image to be imported by simply clicking on ***the Get Image button (1)***. A standard Open dialog is shown where you select a suitable image. Mots commonly used file formats such as PICT, JPEG, BMP, TIFF, TARGA, PDF and so on can be used.

Some elements in the drawing area are: The original ***Image Box(2)*** that encloses the original image. The ***Clip Line Guides (3)*** that forms the ***Clip Area (4),*** which is what is actually exported to the main drawing window. The Clip Area can be resized by clicking on the respective lines and dragging. If the Image Box and Clip Area lines overlap, the Image Box is selected, but you can always get access to the Clip Area controls by using the extended ***Clip Guide (3)*** lines. The Image Box can be deformed by means of the red ***Image Box Deformation handles (5-6)***. When the Image Box is deformed the image follows the box to generate a deformed image, which can be used for removing the perspective from an image to make it easier to use a modeling guide. The ***four corner handles (5)*** are used for free-form deformation. Note that you can generate a horizontally or vertically constrained deformation drag by pressing the Shift key while using these handles. The ***four intermediate deformation handles (6)*** are used for skewing the image by moving one side as a unit. The red lines in between the red control points is used as a ***rotation handle (7)*** for the entire image. When pressing the ***Alt/Opt key***, the rotation only occurs ***on the one side*** you clicked on. The rotation then centers around the red dot in the middle of the selected line, resulting in some sort of image deformation. The ***Bulls Eye (8)*** is used for placing ***a zero point relative to the 3D model*** and you simply drag it to a suitable location. Note that you can fine-tune the location afterwards so if is OK to be a bit crude when placing it. The Measuring Line(9) is used for control measuring elements of the image. You may for example know the distance between two points in the image but not the total width or height of the clip box. You then move the line endpoints to the corresponding location and then enter the true length of the line in the corresponding edit fields below (9) to get a correct scale. Note that the ***Clip Box width and height edit fields (11***), ***the line width and height (9)*** and the coordinates are updated accordingly to enable real scale measuring. The same of course happens if you change ***the clip box height or width values (11)***, where the corresponding values are updated. If the horizontal and vertical dimensions do not correspond, it is possible to adjust the ***proportions*** by means of the ***StretchH and StretchV (15)*** edit fields. They have a default setting of 100 (%) and if you change for example the horizontal filed to 101 (%), the image is stretched by one percent horizontally whereas the vertical proportions remain unchanged. Note that the field resets itself to 100 (%) afterwards to prepare for a new operation. It is also possible to ***rotate the entire image (15)*** by a numerical value, for example to adjust a slightly rotated scanned image. Positive angle values rotate the image counter-clockwise and the units used are in degrees. Before exporting the image to the main drawing window you need to specify the main view to be used using one of the three ***View buttons (16)***.

When the image is correct and ready for use you simply press the ***Done (17)*** button to return to the main drawing window together with the new image.

Some practical suggestions and methods when working with background images. If you find that you need to move the background image a bit or scale it slightly, you can do so by selecting the Edit Background Image command found in the View menu. This displays a floating window where you can nudge the image, move it by entering a numerical value in the edit fields or scale it. Another thing worth noting is that a modified version of the image is stored at the location of the original image, using the same file plus an additional letter stating which view it belongs to (e.g. MyImage -> MyImageF to indicate that it belongs to the Front view. The reason for this is that background images are not stored within the actual TouchCAD file, it only stores the address to it. If you move the file to a new location, TouchCAD finds that it can't find it and therefore asks for help when the file is opened again. If so, just locate it again on your hard disk and it stores the new location or cancel the background image if you don't need it.



Page Setup

The *Page Setup* command is used for extracting a ***page size*** and ***orientation*** from the printer driver and for setting ***how many sheets*** to be used for defining the ***light blue drawing / sheet space area*** found in the drawing area of the main drawing window. Below is a picture showing how the first dialog looks like when selecting ***the Pager Setup*** command.

TouchCAD allows you to create a sheet space over ***any number of sheets***. You can for example join two A4 sheets (the same applies for letter size sheets) into one A3 drawing by placing two sheets beside one another. ***The light blue rectangle*** seen on the screen represents the ***true available drawing area*** of the printer you use. It is therefore easy to join two sheets by simply ***cutting*** off one of the sheets ***along the edge*** of the area actually being used by the printer, and then place it on top of the other sheet. This will generate a perfect match, or at least as good as the printer can manage. The number of ***steps (sheets) horizontally and vertically (1-2)*** simply defines the ***number of sheets to be used.***

The ***Print connection markers checkbox*** adds ***small diagonal lines*** in the corners of all ***sheet***s and helps in reassembling the sheets.

The ***Page numbers check box (4)*** prints a ***page number*** in the corner of each sheet and helps to keep track of where you are on the assembled sheet.

The ***Show Page Breaks*** adds a line ***around each sheet.***

A standard ***Page / Print setup dialog*** occurs after having ***clicked OK*** on the first dialog. Here you specify the printer specific properties.

Print...

The ***Print*** command prints out the drawing on ***sheet*** in a standard way. The quick key is ***Command P (M) / Control P (W).***

Quit...

The Quit command closes down TouchCAD. If the currently active document is unsaved, a dialog occurs wherein you specify if you want to save your work or not. The quick key is ***Command Q (M) / Control Q (W)***. Note that The Quit command is located in the TouchCAD menu on the Mac OSX version.

Recently Used files

The last items in the File menu are used for storing the fine name and location of recently used files. These can easily be opened by using these commands and without having to look for them.

Edit menu

The *Edit menu* contains various editing tools.

Undo and Redo

The ***Undo and Redo*** commands are standard features in most programs. TouchCAD provides ***five Undo/Redo steps*** and operates in a loop. The ***quick key*** for ***Undo*** is ***Command Z (M) / Control Z (W)***. The ***quick key*** for ***Redo*** is ***Command Y (M) / Control Y (W)***.

Copy and Paste

The ***Copy and Paste*** commands are primarily used for ***copying the drawing area*** and pasting the copy ***into the clipboard*** or to ***other programs***. The ***quick key*** for the ***Copy is Command C (M) / Control C (W)***. The ***quick key*** for ***Paste*** is ***Command V (M) / Control V (W)***.

Clear

The ***Clear command deletes*** unwanted objects from the drawing. Other possible methods to delete objects are to use the ***Delete*** or ***Backspace keys*** on the keyboard.

Group and Ungroup

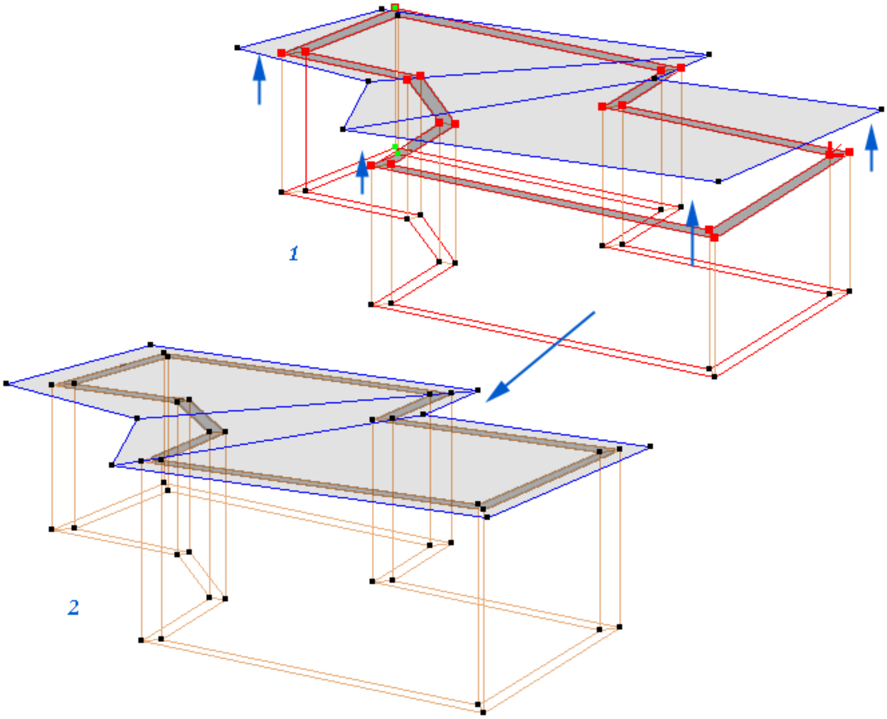
The ***Group*** command is used for ***joining*** one or several ***object(s)*** into one unit.

TouchCAD only supports ***single level groups*** so grouping of for example two groups results in a group, which when ungrouped, consists of a number of individual objects.

Clicking on a grouped object result in a selection of all control points within the group and such an object is therefore treated as a unit.

Many tools in the Tool palette and Tool menu generates grouped objects by default, and you may need to either ***ungroup*** them or use the ***Ignore Group mode*** command to be able to edit the object.

The ***quick key*** for ***Group*** is ***Command G (M) / Control G (W)***. The *quick key* for ***Ungroup*** is ***Command U (M) / Control U (W)***.



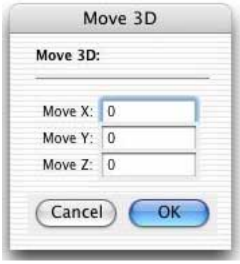
Ignore Groups

The *Ignore Groups* command disables the group identities and enables individual editing without having to ungroup the objects first. A ***check*** before ***the Ignore Group command text*** means that groups are disabled and visa versa.

The picture above shows an example. The yellow walls form a group when drawn as a surface using the Double-line tool. Imagine that you want to project the upper control points to a roof surface. If you click on one of the wall control points, all points will be selected. You could of course ungroup it but in some cases it is easier to use the Ignore Groups mode. In this case it is used to temporarily ***disable the wall group***, select the ***upper control points*** *(1)* and then ***project*** them to the roof surface using the ***Align to Surface tool*** *(2)* and then ***enable groups*** again.

Move...

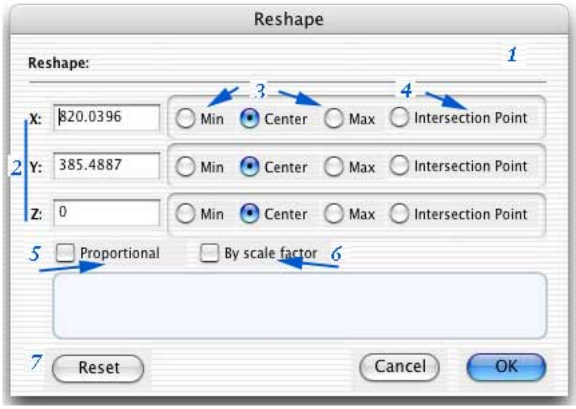
The *Move...* command moves selected objects by means of a dialog box. Enter the relevant offsets in the dialog and click OK.



Other methods for moving objects include entering ***new coordinates*** in ***the edit fields*** above the drawing area, which ***moves the Intersection Point*** to a new location and moves all selected objects the same distance. The ***Intersection Point*** coordinates can also be ***moved relative*** to the previous location by placing an ***"M"*** before the coordinates. Typing M200, for example, means that the current coordinate in increased by 200 units and M-100 means that the coordinate in decreased by 100 units.

Reshape...

The ***Reshape...*** command is used for changing the dimensions of the selected object or objects by means of a dialog box. Below is a picture showing the ***dialog box (1).***



The three ***X, Y and Z (2)*** fields are used for entering new dimensions, or, if the ***By Scale Factor checkbox*** is checked, by entering a scale factor.

The object can be ***scaled*** individually in all three directions from ***the minimum, center, or maximum (3)*** side or from the ***Intersection Point (4).***

The ***Proportional (5) checkbox*** enables you to scale the selection proportionally. If the X dimension was 1000 units and you type in 500, the Y and Z dimensions are automatically halved. This can be quite practical if you want to produce a series of more or less identical objects that only vary in physical size.

The ***By Scale Factor (6) checkbox*** enables you to scale the selection by means of a scale factor instead of using actual dimensions. The default is one and if you multiply a coordinate by one you will get the same value, that is no scaling. Entering for example 0.5 will halve the size and 2 double it and so on.

The ***Reset*** *(7)* button is used for resetting the dimensions to what you had when the dialog first opened. Quite practical when you lost track of what you where doing.

The ***Quick Key*** for the ***Reshape*** command is ***Command R (M) / Control R (W).***

Other methods for ***resizing*** objects include ordinary ***dragging*** on the screen, using the ***Resize box*** on the screen and using the Resize features found in the ***Dimension Tab*** of the ***Object Info*** palette.

Rotate ->

The ***Rotate-> sub-menu*** items allows you to rotate points and objects in various ways.

Rotate -> Left 90 degrees

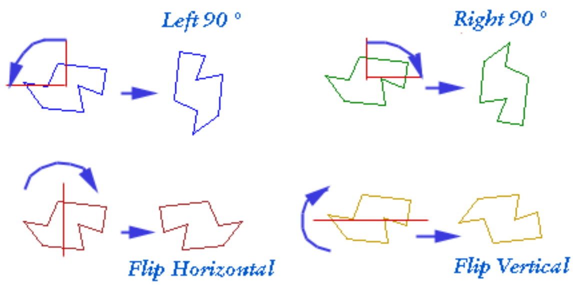
The ***Rotate -> Left 90 degrees*** rotates selected objects 90 degrees to the left (counter-clockwise) relative to the current view. The ***Quick Key*** for ***Rotate -> Left 90 degrees*** is ***Command L (M) / Control L (W).***

Rotate -> Right 90 degrees

The ***Rotate -> Right 90 degrees*** rotates selected objects 90 degrees to the right (clockwise) relative to the current view.

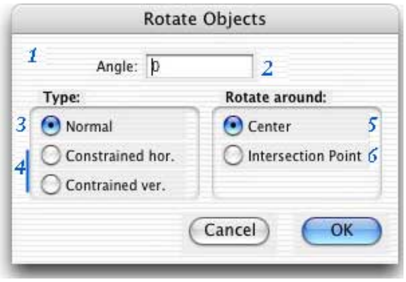
Rotate -> Flip Horizontal / Vertical

The ***Rotate -> Flip Horizontal / Vertical*** commands are used when mirroring the selected objects horizontally or vertically ***around*** the physical ***centerline*** of the selection.



Rotate -> Rotate...

The Rotate...command rotates the selected objects by means of ***a dialog box (1)***. Below is a picture showing the options.



The rotation ***angle*** *(2)* is entered in the edit field at the top.

The ***Normal*** *(3)* rotation method rotates the selection in the ordinary way.

The ***Constrained Horizontal and Vertical methods*** *(4)* block out the calculation in one direction and therefore results in a skewed and distorted result which might be useful in some cases. These methods work in the same way as found in the ***Mode*** bar when using the ***Rotate*** tool in the ***Tool*** palette and we therefore suggest that you study this section of the manual to understand these methods.

The rotation can center around the physical ***center-point*** *(5)* or on the ***Intersection Point*** *(6)*.

Other methods for rotating objects and selected points include using ***the Rotate and Mirror tools*** in the ***Tool palette***.

Select All

The *Select All* command selects all objects that can be accessed with the current layer setting. Invisible objects are ignored. The ***Layer visibility*** command found in ***the Page menu*** and as a pop-up in the ***Layer tab*** of the ***Layer/Color palette*** also affects the behavior of the Select All command. The Show / Snap /Modify Others mode is the only mode allowing the selection of objects located on other layers.

The ***quick key*** for the ***Select All*** command is ***Command A (M) / Control A (W)***.

Select All Points ->

The ***Select All Points*** sub-menu allows you to improve a selection by selecting more relevant points. In most cases you start by selecting a single point and then use these commands to select the rest of the relevant points. Note that most of these items are also available in the contextual menu displayed when you have a two-button mouse or by pressing the ***Control key*** before clicking (Mac with a one-button mouse).

Select All Points -> In Object selects all control points in an ***object [command + 5 (M) / control + 5 (W)]***.

Select All Points Along Ruler selects all control points along the currently selected control ruler ***[command + 6 (M) / control + 6 (W)]***.

Select All Points Perpendicular to Ruler selects all control points perpendicular to the currently selected control ruler ***[command + 7 (M) / control + 7 (W)]***.

Select All Points Along the Edge selects all control points along the currently selected surface edge *[command + 8 (M) / control + 8 (W)]*.

Custom Selection ->

The *Custom Selection sub-menu* selects objects having specific properties. A typical use is to select for example all points used as design aids during the design process and then delete them. The options are:

Custom Selection -> Points selects all objects consisting of a single control point.

Custom Selection -> Rulers selects all objects consisting of a single control ruler.

Custom Selection -> Curves selects all objects consisting of a single curve (a series of ruler starting points).

Custom Selection -> Rulers Curves selects all objects consisting either of a single control ruler or curve.

Custom Selection -> Surfaces selects all objects defined as being a surface, that is having at least two control rulers and at least two control points on one of the rulers.

Custom Selection -> In Current Layer selects all objects belonging to the currently selected layer.

Custom Selection -> In Current Color selects all objects belonging to the currently selected color.

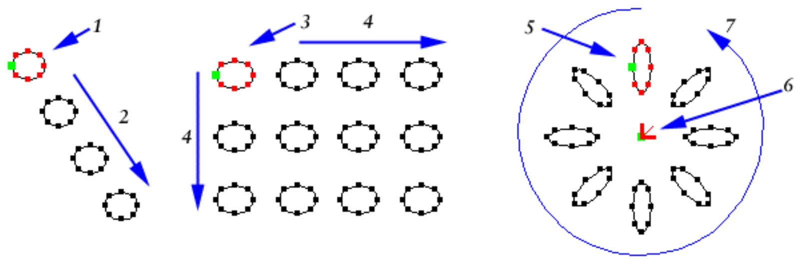
Duplicate

The *Duplicate* tool is used for creating copies of objects. The *quick-key is [command + D (M) / control + D (W)]*

Another way of creating a copy of an object is to *select* it *using* the *Object Selection* mode in the Selection mode bar, *click* on the object *while pressing the Opt (M) / Alt / Opt (W)* key and drag. This generates a copy of the selection.

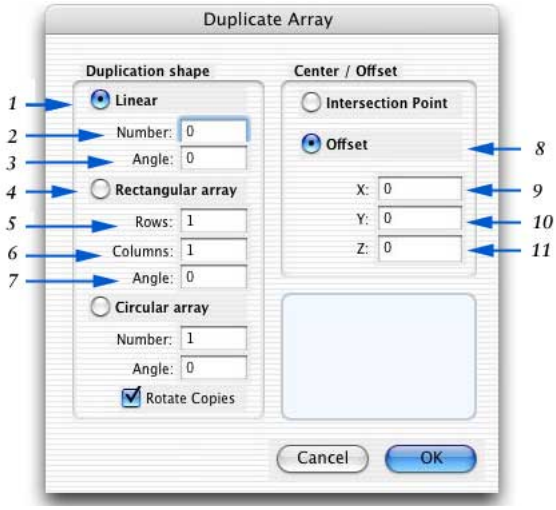
Duplicate Array

The *Duplicate Array* command found in the *Edit* menu is used for adding several copies of the selected object(s). Objects can be duplicated in three different ways; by *fixed offsets* (*1-2*) in the X, Y and Z directions, by *offsets in rows and columns* (*3-4*) or by *rotating and copying objects (5-7)* around a fixed *center-point* (*6*).

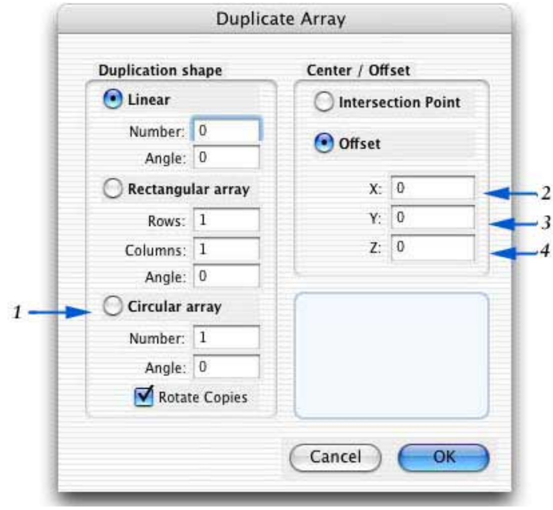


In the two first cases you select the objects to be copied and then run the *Duplicate Array* command. In the *circular* case you have the option to manually enter the coordinates of the center-point, but in most cases it is easier to use an object as the center-point. Placing the *Intersection Point* on a special *center-point object* (*6*) before running the ***Duplicate Array*** command does this. This object is ignored by the command.

The ***Duplicate Array*** dialog can be seen below.

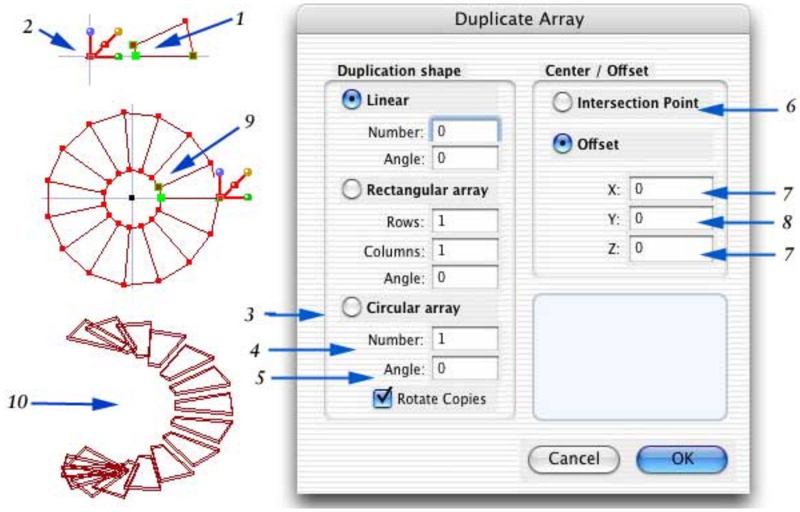


1/ Linear mode
2/ Number of copies
3/ Rotation angle for each new copy
4/ Copy in rows and columns
5/ How many copies in the row direction (rows and columns are always oriented as seen on the screen)
6/ How many copies in the column direction
7/ Rotation angle for each new copy
8/ Offsets is the only available option in the Linear and Rectangular modes
9,10,11/ Offset distances for each new copy.



The offset fields works in a slightly different way in the ***Circular mode*** (*1*). Two of the coordinates are used as center-points (*2-3*). The third field is used for in the direction away towards you (*4*). Note that the direction is determined by the view used.

The offset can for example be used to generate the steps in a stair as seen below.

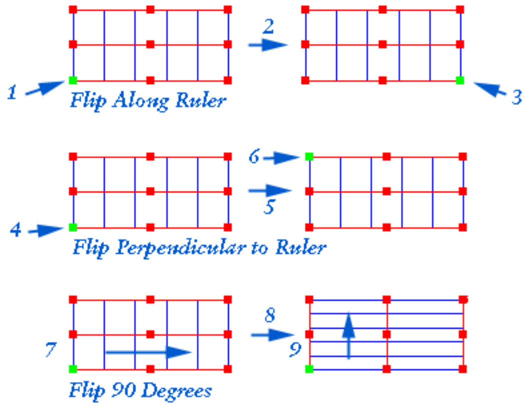


Select the ***object (1)***, and then the ***center-point object (2)*** to place ***the Intersection Point*** in the middle. Chose ***Duplicate Array*** in the ***Edit*** menu. Select the ***Circular array*** (*3*) mode. ***Fourteen steps (4)*** were used in this case, the first ***selected objects not included***. Set the ***angle (5)*** between each copy. Click on the ***Intersection Point*** (*6*) button. Leave the two ***center-point (7) coordinates*** unchanged. Add a ***pitch*** offset in the ***Vert. Offset*** (vertical) field (*8*). Click OK, and the ***stair (9)*** is generated. The steps as seen from an ***isometric view (10).***

Flip Index Direction->

The *Flip Index Direction* commands change ***internal index order*** of the ***control rulers along or perpendicular to the rulers***, and ***the direction of the rulers***.

One fairly common example of use would be if you want to add more controls downwards and towards the green starting dot. In such as case you simply flip the direction along the rulers and then add point upwards as usual. Below is a picture that illustrates the steps.



The *Flip Index Direction Along Ruler (2)* command moves the green starting dot from the ***left side*** *(1)* to the ***right side*** *(3)*. In the ***left*** *(1)* case, the first control point is located to the ***left*** whereas in the ***right*** *(3)* case it starts to the ***right***.

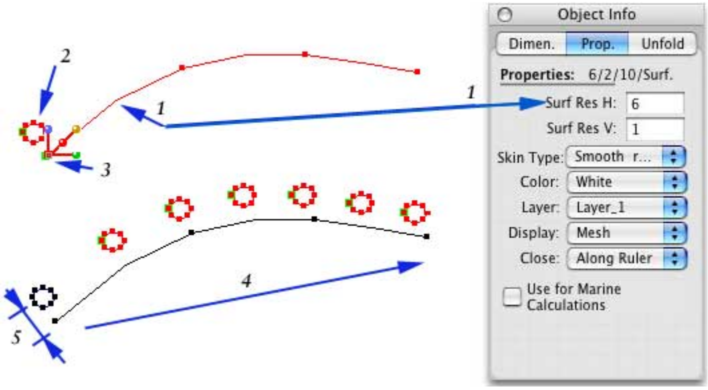
The *Flip Index Direction Perpendicular to the Ruler (5)* command moves the green starting dot from the ***bottom side*** *(4)* to the ***upper side*** *(6)*. In the ***left*** *(4)* case, the first control ruler is located at the ***bottom*** whereas in the ***right*** *(6)* case it is located at the ***top***.

The *Flip Index Direction 90 Degrees (8)* command rotates the ruler direction from being ***horizontal*** *(7)* to being ***vertical*** *(9)*. The green starting dot is still in the same place and has therefore not moved.

The Flip Selected Ruler and Curve commands flip a single element within a surface object. An example of use is when a surface is generated using the Loft command abd where one of the rules gets a flipped direction by mistake. It is then quicker to just fix it with these commands instead of taking it apart to re-do it again.

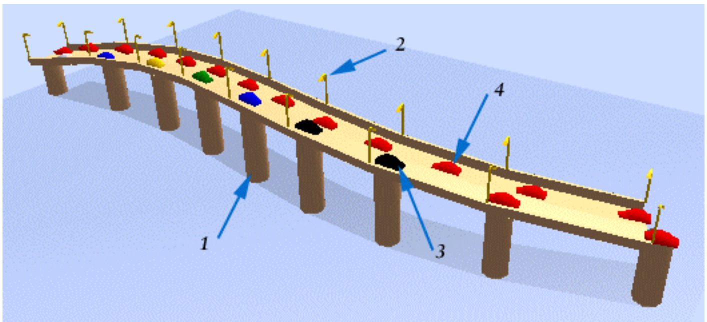
Duplicate Along Ruler

The ***Duplicate Along Ruler*** command ***duplicates*** objects ***along a control ruler***. It can for example be used for duplicating flagpoles or trees along a road, guard rail poles along the sheer line of a boat, etc. The picture below shows the steps.



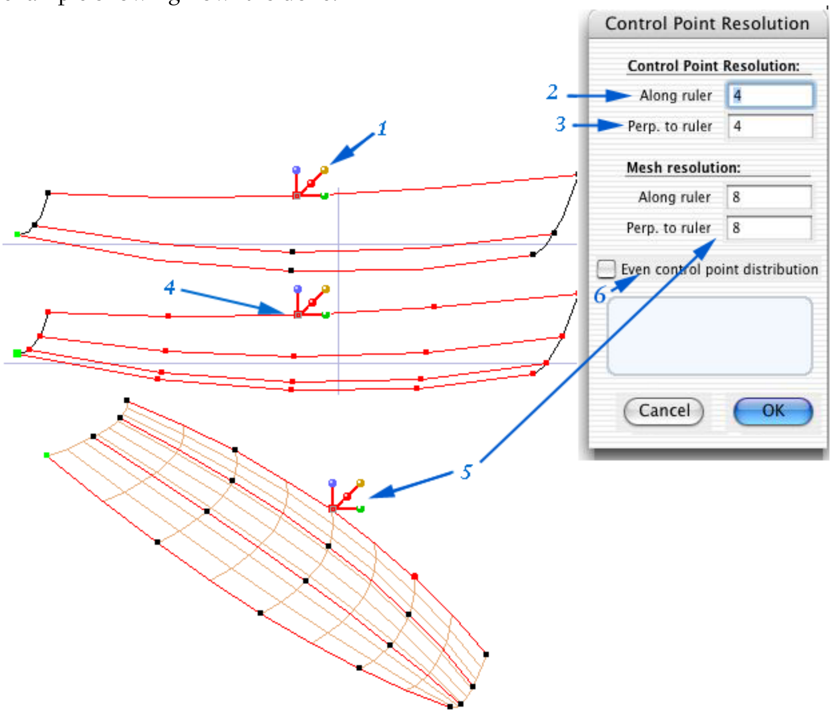
Start by preparing a ruler. The number of copies is controlled by the number of steps along the ruler and is set in the ***Object Info*** (***1***) palette. ***Select*** the ***object*** or ***objects*** ***(2)*** that you want to duplicate and then shift-click on the ***control ruler*** ***(3)*** to be used. The object that contains the Intersection Point is ignored in the duplication process and is used as the extrusion path in the calculation. ***Select Duplicate Along Ruler*** in the ***Edit*** menu. The result shows up in the lower picture (***4***). Note that the ***relative distance*** from the ***Starting point*** of the path curve is ***preserved***, so it is a good idea to ***place*** the ***objects*** to be ***duplicated relative to the starting point*** of the curve. Any surface ruler can be used. Just place the Intersection Point in the relevant curve by clicking (or Shift-clicking on it).

The picture below shows several examples where the ***Duplicate Along Curve*** command has been used. The ***bridge columns (1)***, the ***light poles (2)***, the ***cars (3 & 4)***. Note that the traffic is heavier in one direction and this was achieved by temporarily ***changing*** the ***resolution*** along the ***bridge*** slightly before duplicating. This works because the ***number of duplicates*** is controlled by the mesh ***resolution*** in the ***control object***.



Control Point Resolution

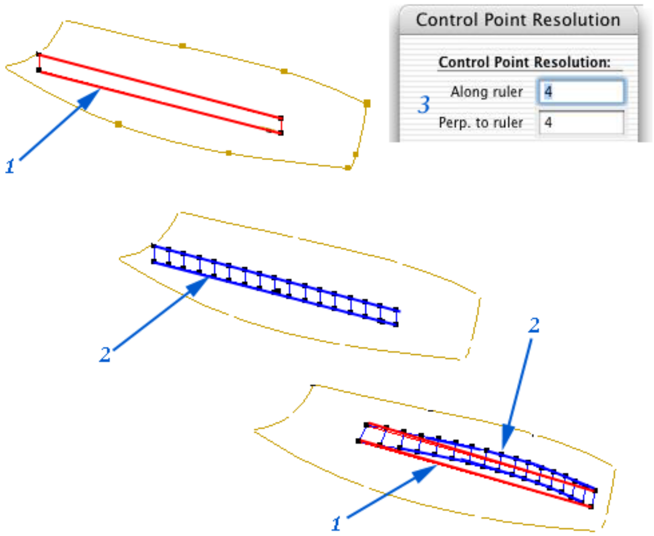
The *Control Point Resolution* command ***adds*** or ***deletes control points*** to and from an object. This can for example be useful when you suddenly realize the need for more control points to control the shape of a curve or surface. You can of course add controls manually but this method usually ***maintains the basic shape*** well. Below is an example showing how it is done.



Click on the ***object*** *(1)* to be modified. ***Choose Control Point Resolution*** in the ***Edit*** menu. Change the ***Control point resolution*** *(2-3)* to for example four by four and the mesh resolution (5) to say eight by eight. Note that the ***new points*** *(4-5)* where added along the shape of the original surface. The ***mesh resolution*** *(5)* was also changed, which essentially the same as in the Object Info palette though done in a single step.

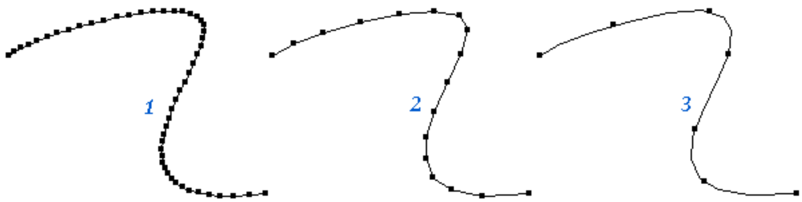
The ***Even control point distribution*** *(6)* checkbox does the same as ***the Redistribute Controls*** command though it is done in a single step and applies to all rulers in the object.

Below are a few practical examples showing the use of the ***Control Point Resolution*** command.



Imagine that you want to project a ***surface (red / 1)*** to the yellow surface using the ***Align - > Surface*** tool. If you do it with a resolution of ***1 x 1*** you will get a very poor alignment to the surface (third picture). If you instead increase the resolution to say ***16 steps along the ruler*** using the ***Control Point Resolution (3)*** command and then the ***Align -> to Surface*** you will get a much better result ***(blue / 2)***

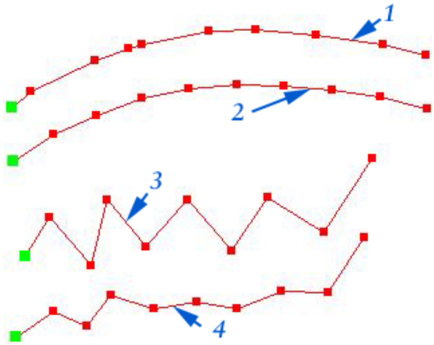
The ***Control Point Resolution*** command can also be used ***for reducing the***



resolution. Imagine that you have imported a polygon having for example fifty ***control points (1)***. A ***recommended rule*** when working with math is to ***use as few control points as possible*** without loosing the shape. In such as case, just ***decrease*** the resolution to say ***16 steps (2)*** by using the ***Control Point Resolution*** command. As you can see in the second picture, the shape is still almost identical to the first picture. Even a reduction to ***eight control-points (3)*** with a curve resolution of ***sixteen steps (3)*** works in this case. If done correctly, you will end up having the same shape, but with a manageable number of controls, and it is quicker too. Experiment!

Redistribute Controls

The Redistribute Controls command is used for creating an even distance between the control points. An example of use is to clean up and generating smoother and more harmonious objects. A more even distribution typically generates better looking and less nervous math curves.



The diagram above shows some examples. The ***upper ruler curve*** *(1)* forms a smooth looking shape though the control points are not evenly distributed. The ***second ruler curve*** *(2)* looks the same but the controls are more evenly distributed.

The Redistribute Controls command does not always do a good job and should therefore be used with some caution. The third example (3) shows a very uneven shape and when the Redistribute Controls command here generates a fairly different shape. In other words, the Redistribute Controls command works bet on basically smooth looking shapes.

Worth noting is that the ***Redistribute Controls command works on the selected object*** but only on control rulers where at least one control point has been selected. In other words, it is possible to redistribute one, some, or all control rulers within a given selected object.

Join Objects

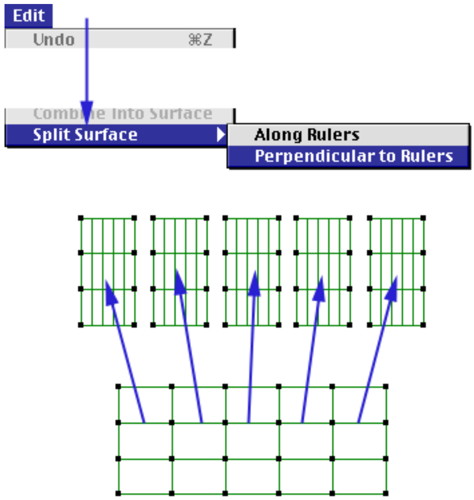
The ***Join Objects*** command is used for ***joining two objects into one***. It works in a similar way to the ***Join tool*** in the ***Tool*** palette, but instead of drawing a joining line along the edge you simply ***select the two objects*** to be merged and then ***run the Join command.***

Two objects can be joined provided that they have the same number of control rulers and that they are physically connected along one edge and that the rulers have the same direction. Surfaces as well as curves and control rulers can be joined.

The ***quick key*** are *[command + J (M) / control + J (W)].*

Split Objects

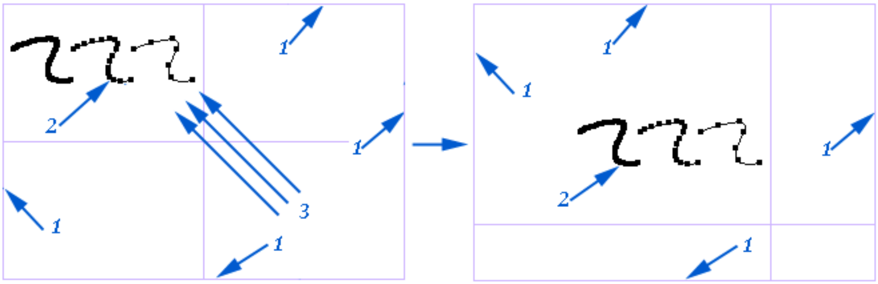
The *Split Surface* commands split objects into ***strips, along or perpendicular*** to the ***control rulers***. Just place the ***Intersection point*** on the object and then select the command. The picture below shows the result of a ***perpendicular*** split. Cutting along the rulers works in the same way but in the other direction.



Objects can also be divided into separate objects using the ***Cut tool*** in the ***Tool*** palette. This tool allows individual cutting along a give cutting line, which is not the case with the Split Object commands.

Page menu

The *Page* menu contains various document settings such as the location of the ***sheet*** area, the degree of zoom, the layer visibility, drawing scale, units and grid, cross-sectioning settings, and origin settings.



Move Print Area

The Move Print Area is used for moving the intended ***sheet*** area relative to the model. In the picture above the ***objects*** *(2)* are located in the upper left corner of the ***sheet area*** *(1)*. Select the ***Move Print Area command*** and ***drag*** *(3)* the ***sheet*** to a suitable location around the objects.

Note that ***each view*** in the View bar, above the drawing area, has its ***own individual setting*** and requires its own individual adjustment.

Fit to Window

The ***Fit to Window*** command adjusts the degree of zoom and pan settings so that the ***sheet*** area fits into the drawing area. This is an extensively used command by nearly all users so it is a good idea to memorize the ***quick key*** ***[command + 4 (M) / control + 4(W)]***.

The ***Fit to Window button***, in the ***bottom left corner*** of the drawing window does the same thing as the Fit to Window command in the Page menu. Just click on it.

There are also other ways to adjust the degree of zoom. The Zoom and Pan tools in the Tool palette. The View buttons in the bottom left corner of the drawing area. Pressing the Space bar key temporarily activates the Pan tool. Changing view automatically centers the pan on the objets selected.

Layer Visibility->

The *Layer Visibility* commands work in the same way as the ***pop-up*** menu in the *Layer* / Color palette. They enable you to control what you see in the drawing area.

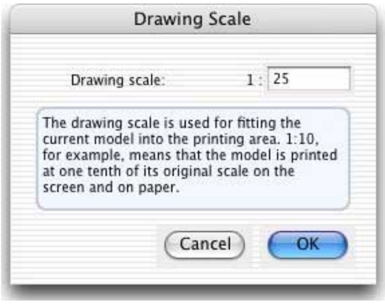
The four layer ***visibility modes*** are:

Active Only wherein you only see the active layer, regardless of the fact that other layers are visible or not.

Show All displays all visible layers but does not allow snapping to objects in other layers.

Show / Snap Others displays all visible layers and allows snapping to object in other layers but not editing them.

Show / Snap / Modify displays all visible layers and allows both snapping and editing of visible layers.

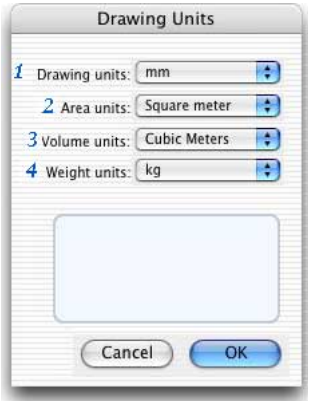


Drawing Scale

The *Drawing scale* command ***scales*** the model into something that fits ***within the sheet space*** boundaries. A house may for example need a scale of 1:100 to fit into the ***sheet*** space, whereas a chair would fit in at a scale of say 1:10.

It is also possible to ***enlarge*** the model to fit into the ***sheet*** space by entering a value smaller that one. A scale of 1 : 0.5 is the same as a drawing in scale 2:1 and 1 : 0.25 equals 4:1.

Note that the ***drawing scale*** does ***not change the physical size*** of the model. Changing the drawing scale also changes the default setting used when opening a new blank document.



Drawing Units

The ***Drawing units*** command is used for defining the drawing units used in the document. There are four different settings in the dialog. The reason for this is that it may be practical to use for example millimeters for measuring lengths but not for measuring areas or volumes.

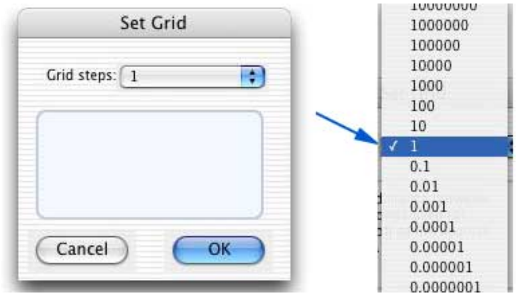
The ***Drawing Units*** *(1)* define the ***length*** units used by TouchCAD, and the options are mm, cm, dm, meters, decimal inches and decimal feet.

The ***Area Units*** *(2)* options are mm, cm, dm, meters, decimal inches, and decimal feet.

The ***Area Volume*** *(3)* options are mm, cm, dm, meters, decimal inches and decimal feet.

The ***Weight Units*** *(4)* options are kilos, metric tons and lbs.

Note that changing the drawing units also changes the default setting used when opening a new blank document.



Set Grid, Change Grid Resolution->

The ***Grid resolution*** command controls several things. It is used for controlling the number of ***decimals displayed*** when drawing. Note though that it does not affect the true accuracy of the program, only what you see. A second use is to control the ***step length*** used ***when nudging*** objects with the arrow keys.

The ***Set Grid*** command displays a dialog wherein you can choose between a number of options in a pop-up.

The *Change Grid Resolution -> Bigger Steps* command ***increases the grid steps*** used by a factor of ten (e.g. 1 -> 10 -> 100 -> 1000). The ***quick key*** for this command is *[command + 2 (M) / control + 2 (W)].*

The *Change Grid Resolution -> Smaller Steps* command ***decreases the grid steps*** used by a factor of ten (e.g. 1 -> 0.1 -> 0.01 -> 0.001). The ***quick key*** for this command is *[command + 1 (M) / control + 1 (W)].*

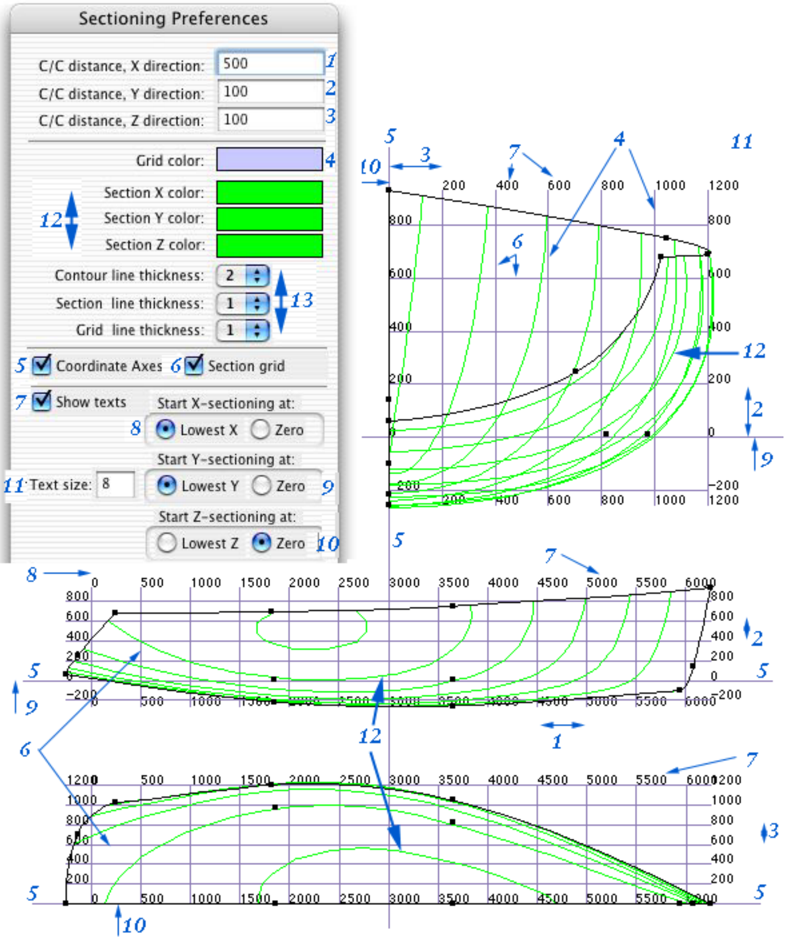
An easy way to remember which is which of these quick key commands is to remember that *2 is bigger than one (bigger steps) and 1 is smaller than 2 (smaller steps).*

Text Sizes

The Text Sizes command controls the size of texts in the 3D view as well as in the Unfold view. They are defined in standard point sizes in the dialog that follows when the command is selected.

Sectioning Settings

TouchCAD allows you to work with ***dynamic line drawings***, which automatically update when you move control points. The ***Sectioning Settings*** command controls the behavior of surfaces using the ***Cross Sections display mode.*** Any surface can be used and you can ***change display mode*** of a surface at any time using the ***Display mode pop-up*** found in the ***Properties tab*** of the ***Object Info*** palette.



The ***Sectioning Settings*** command allows you to specify what kind of cross sectioning you want to see, the distances between the cross sections and so on. Below is a picture showing the settings dialog.

The ***c/c distance (1-3) X, Y and Z*** fields specify the sectioning distances used in the current document. The settings are individual for each direction.

The ***Grid Color (4)*** setting allows you to change the color of the grid by clicking on the color field. A standard color dialog occurs where you can choose any color.

The ***Zero Lines (5)*** checkbox allows you to see the standard horizontal and vertical zero lines.

The *Sectioning Grid (6)* checkbox activates the display of the sectioning grid.

The ***Show Texts*** *(7)* checkbox controls whether you will see the sectioning texts or not.

The ***Start Sectioning At X, Y, Z,*** *(8-10)* controls where the sectioning lead texts place their respective reference points. In the example the Y direction places the zero point at the waterline and all reference lines have a negative value. In the X direction, the zero point is set at the leftmost section that fits into the given surface and current c/c sectioning setting. TouchCAD automatically updates the texts if the surface is made longer in such a way that more sections can be added.

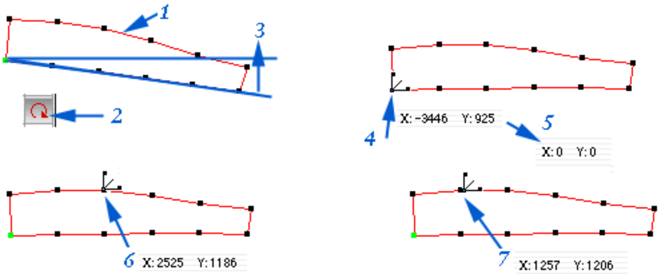
The *Text Size (11)* field allows you to specify the size of the lead texts.

The *Section Color (12)* settings define the color of the cross section lines in the X, Y and Z directions.

The *Contour, Section and Grid line* thickness *(13)* pop-ups define the thickness of the respective lines when printed. It is used for making a more clear distinction between the line types and make a drawing more clear when printed in a large formats.

Set Origin, Reset Origin

In many cases it can be useful to get the coordinates from a given point instead of from the fixed coordinate system. It allows you to get dynamic offsets from a given point without having to calculate an offset. The ***Set Origin Reset Origin commands*** are used for creating a ***temporary offset*** in the dynamic coordinates shown above the drawing area in the drawing window. It does not really move the coordinate system and does not affect the Intersection Point coordinates. TouchCAD provides ***two offsets***, one for the *3D-views* and one for the *Unfold view* and you *set and reset* them *individually*. The quick keys for *Set Origin* is *[command + 9 (M) / control + 9 (W)]* and *the Reset Origin* quick keys are *[command + 0 (M) / control + 0 (W)].*



The ***unfolded panel*** *(1)* above must be fitted into a material width of 1200 units. To check it, select the ***Rotate tool*** *(2)* in the ***Tool*** palette and rotate it so that the bottom points form a horizontal line. Select the *Reset Origin* command and click on the ***bottom left corner*** *(4)* of the object to place a ***temporary zero point*** *(5)*. Hold the cursor over the ***control points*** *(6-7)* to locate the ***highest coordinate*** *(7)*. In this case, the Y max was 1206, which probably makes it possible ***to fit it into the given material width*** by ***modifying the 3D shape*** slightly. ***Reset the Origin*** when you're done using the Reset Origin command.

This is an example showing how to integrate an efficient use of material in the design process. In fact, it is good design to make things efficient to produce.

Tool Menu

The Tool menu contains tools for modeling and calculations. Some of the commands are menu version of the tools in the tool palette, used where you essentially want to come to the dialog without having to select a tool in the Tool palette and do something on the screen.

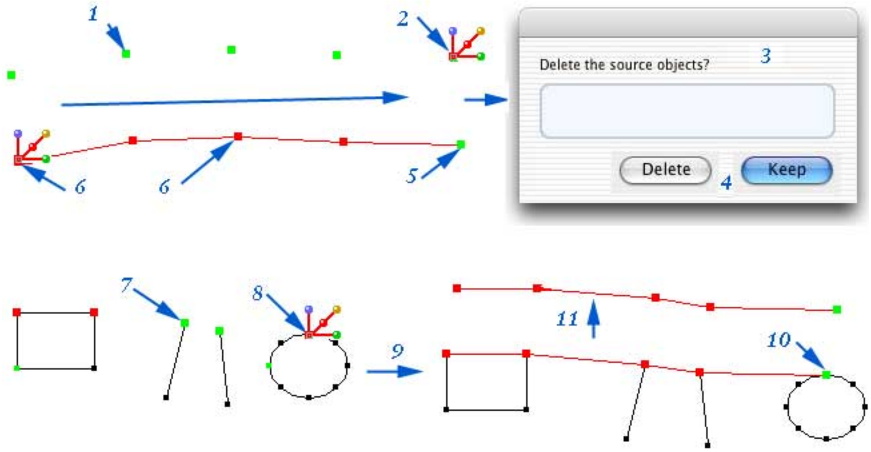
Add Control Point / Ruler

The *Add Control Point command adds* one ore several ***control points*** to a selected object or objects. It essentially does the same thing as can be done by pressing the ***Opt/Alt key*** before clicking to start a dragging sequence. Control points are added up and ***away*** from the ***green starting point*** dot found on all objects.

The *Add Ruler command adds* one ore several ***control rulers*** to a selected object or objects. It essentially does the same thing as can be done by pressing the ***Shift + Opt/Alt keys*** before clicking to start a dragging sequence. Control rulers are added up and ***away*** from the ***green starting point*** dot found on all objects.

Combine into Curve

The *Combine into Curve* command is used for combining a number of control points into a new separate ruler object. The control points can be separate points, or elements in curves, rulers or surfaces. The ***Combine into Curve*** command uses the ***physical distance*** between the selected points when deciding the extrusion path. The ***starting point*** is where the *Intersection Point* is located.



Select a number of ***points*** *(1)*. ***Shift-click*** *(2)* on the ***starting point*** if it is not where you want it to be. Run the ***Combine into Curve*** command. The ***Delete Source*** *(3)* objects dialog occurs where you can choose to ***keep*** *(4)* the original objects (points) or ***delete*** *(4)* them. In the upper example we probably want to delete them to avoid having redundant objects whereas in the lower example it may be better to keep them. The

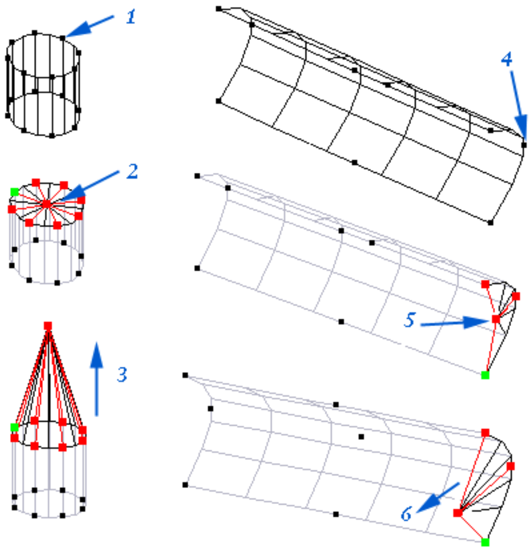
result *(5-6)* is a control ruler with the ***starting point*** *(5)* where the ***Intersection point*** *(2)* used to be.

The lower example shows some selected points being located in various ***shapes*** *(7)* and the ***Intersection Point*** *(8)* in one corner. The result is of course an individual ***control ruler*** *(11)* with the ***starting point*** *(10)* where the ***Intersection point*** *(8)* used to be. In the lower example it was better to ***keep*** the original objects *(4)*.

The most common use for the Combine into Curve command is probably to clean up imported data by creating ruler objects that can be lofted into surface objects. Imported objects such as points or triangles are not very useful in TouchCAD, because it is really based on math based surface objects, but it can be a good base for re-skinning new objects.

Add End Caps

The *Add End Caps* command adds an end surface on an open / uncovered surface end. It only works on control ruler ends. Below is a picture showing some examples.

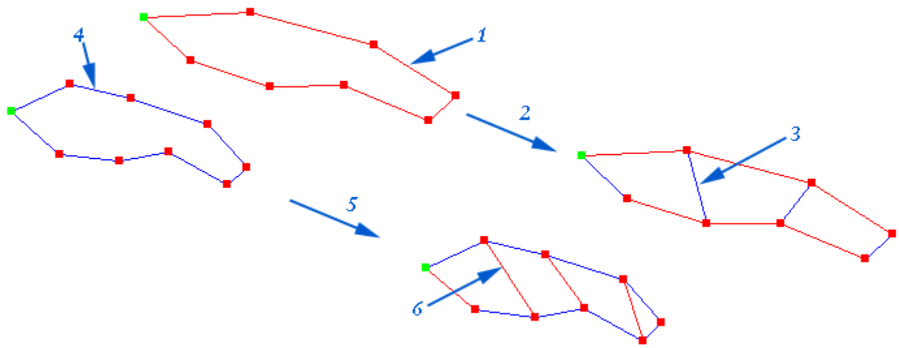


The first example is a cylinder with no end caps. Select all control point along the ***edge*** *(1)*. Run the *Add End Caps (2)* and an ***end surface*** is ***added*** automatically. The cap can be edited by for example dragging the ***center-point*** *(3)*.

The ***second example*** *(4)* is an open surface where you select all ***end points*** *(4)*. Run the *Add End Caps (5)* and an end surface is added automatically. Note that the center-point is not in the middle of the distance between the edges of the original surface. Instead it is by default placed at an average coordinate of all the selected points in the respective directions. It can however be adjusted by ***dragging the center-point*** *(6)* in any direction.

Curve to Surface

Curves (a series of ruler starting points) and ***control rulers*** are ***not surface objects*** and are therefore not visible when rendered and can not be used for unfolding. The ***Curve to Surface*** command can be used for converting ***rulers*** and ***curves*** (ruler starting points*)* into ***surfaces***.



In the first example we have a ***control ruler (1)*** that we want to convert into a surface. You can see that it is a ***ruler*** because ***all lines are red between the control points*** even though this particular object is blue. You can see the color when the object is deselected. Select it and then run the ***Curve to Surface (2)*** command to convert it into a ***surface (3)***. Note that you can see that it is a ***surface*** because it ***has two control rulers*** and ***blue intermediate lines*** (the color of the object).

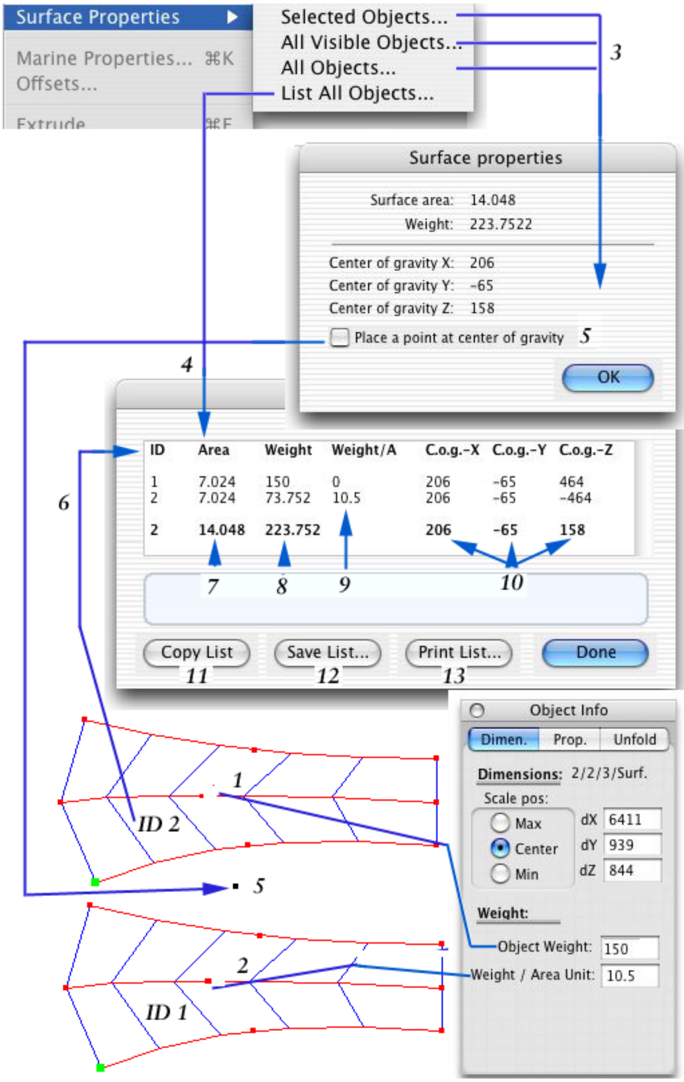
In the second example we have a ***curve (4)*** (a series of ruler starting points) that we want to convert into a surface. You can see that it is a ***curve*** because none of the lines are ***red between the control points*** even though it is selected. Select it and then run the ***Curve to Surface (5)*** command to convert it into a ***surface (6)***. Note that you can see that it is a ***surface*** because it has five ***red control rulers*** and ***blue intermediate lines*** (the color of the object).

Surface to Outline

The ***Surface to Outline*** can essentially be described as a reversed version of the Curve to Surface command. There are however some differences. The Surface to Outline command always generates a new object and this object is always a control ruler.

Surface Properties

TouchCAD provides a range of tools for ***weight, surface area*** and ***center of gravity*** calculations.



1/ Specifies the ***total object weight*** of the object regardless of the surface area. This property is primarily used for fixed objects where the surface area is irrelevant (e.g. engines, batteries, tanks, fittings, etc.). Weights can be added or attached to any type of objects including curves and points, so there is no need to draw a complex model in order to use an engine in a weight and center of gravity calculation.

2/ Specifies the ***weight per surface area*** of a specific object (e.g. 0.3 kg per square meters or 12 pounds per square feet). TouchCAD uses the area and weight units settings specified in the ***Drawing Units*** setting in the ***Page*** menu. Note that this setting requires a surface to generate a weight.

3/ Calculations are initiated by selecting one of the four ***Surface Property*** options found in the ***Tool*** menu. The first three options allow you to calculate the ***Selected Object(s)***, the ***Visible Objects*** or ***All Objects*** in the drawing. A dialog window occurs specifying the combined weight, surface areas and center of gravity of the selected objects. It also allows you to place a ***point*** (*5*) at the ***center of gravity*** of the ***combined center of effort*** of all objects used in the calculation. Such a point is useful for example when you want to compare the center of gravity of the objects with the center of buoyancy of a boat hull.

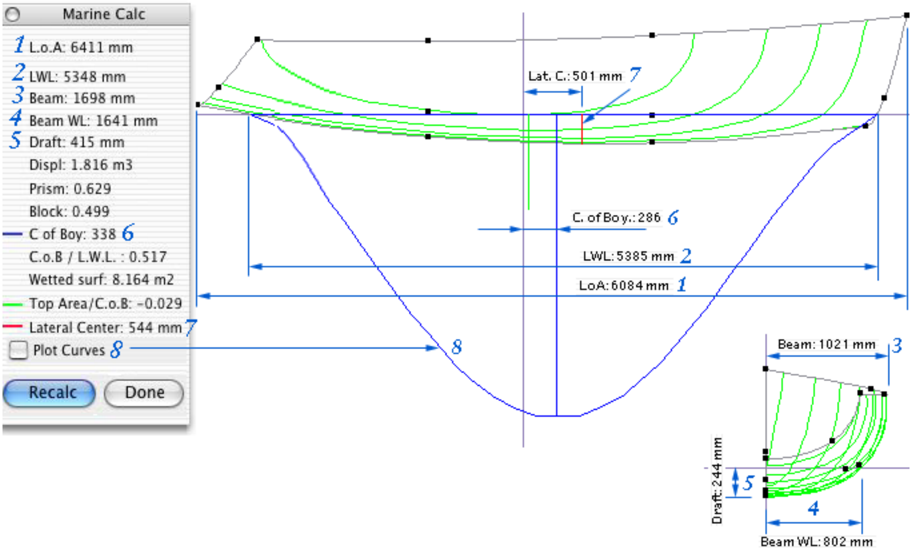
4/ The ***List All Objects*** option in the ***Surface Property*** menu displays a ***spreadsheet*** with the all objects in the drawing ***listed (6-10).*** This dialog allows you to ***copy*** (*11*) the spreadsheet into the clipboard and ***paste*** it into other programs, ***save*** (*12*) it as a ***TAB based text file*** or ***print*** (*13*) it.

Marine properties

TouchCAD provides special features for automatically ***calculating various marine properties*** such as ***wetted surface***, ***prismatic*** and ***block coefficients, section area curves***, ***displacement***, ***center of buoyancy***, ***hull and waterline lengths***, ***total and waterline beam***, etc. The marine calculation property is normally turned off by default, so we therefore need to activate it.

Start by clicking on the hull surface. Select the ***Properties*** tab (***Prop.***) in the ***Object Info*** palette. Click on the ***Marine Calculations*** check box to enable marine calculations on this surface (any number of surfaces can be used though it increases the execution time considerably).

The next step is to select the ***Marine Properties*** command in the ***Tool*** menu. The ***Marine Calc*** palette occurs showing the properties of the surface.



The following objects are displayed in the palette.

Length overall *(1)*.

Waterline length *(2)*.

Beam *(3)*.

Beam on the waterline *(4)*.

Draft of the hull *(5)*.

Center of buoyancy *(6)*, which of course should be balanced against the center of gravity to get the correct floating position.

The ***Lateral center*** *(7)* creates a balance between the lateral center of the hull and the sails. The ***lateral center*** of the ***sails*** can easily be ***calculated using*** the ***Surface Properties*** command in the ***Tool*** menu. On mono-hull sailboats you typically place the center of the sail-plan slightly ahead of the lateral center of the hull to compensate for the asymmetry of the hull when heeling. It can also be useful when positioning the keel /centerboard/ rudders.

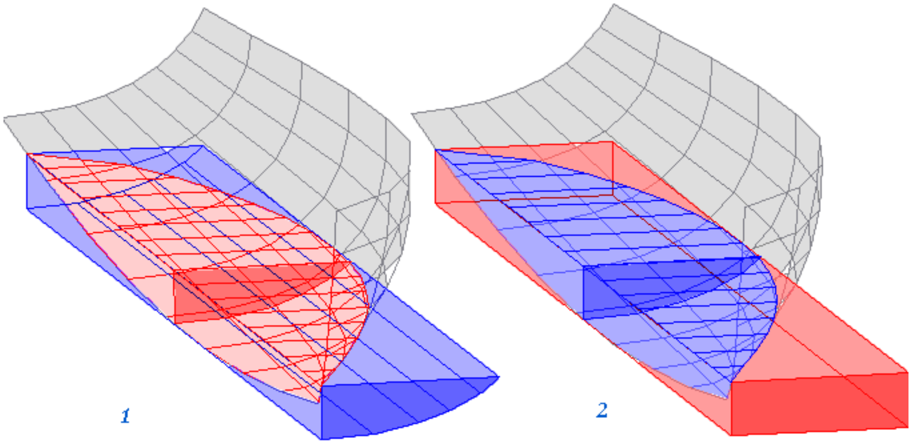
The ***Curve of Areas (8),*** and various other curves can be used dynamically by making small alterations and then running the ***Marine Properties*** command again without closing the ***Marine Calc*** palette.

If you like, you can plot the final curves by checking the ***Plot Curves*** checkbox. The plotting takes place when you close the palette by clicking on the ***Done*** button.

The Prismatic Coefficient (1 below) and Block Coefficient (2 below) are used for estimating the properties of the hull. The coefficients are defined as follows:

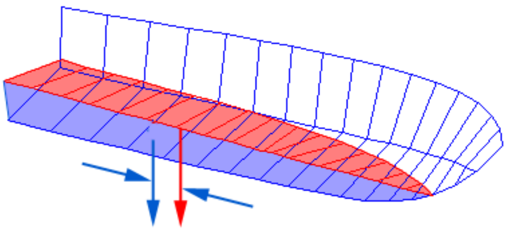
Prismatic Coefficient (1) = ***Volume*** (red) / (***maximum cross-section area*** x ***length in waterline***) (blue)

Block Coefficient (2) = ***Volume*** (blue) / (***cross-section width*** x ***cross-section height*** x ***waterline length***) (red)



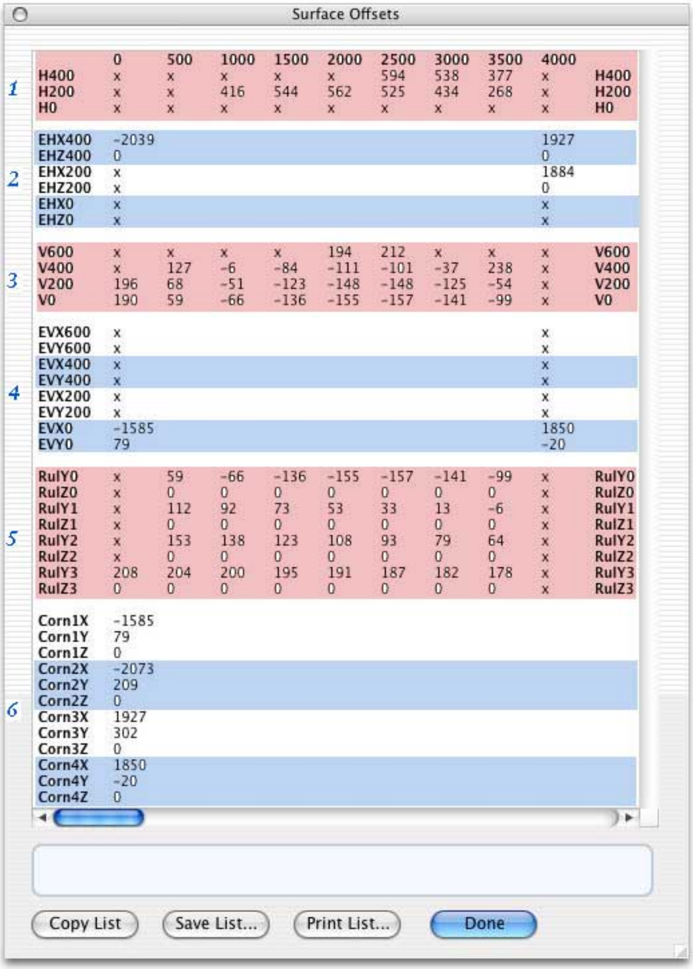
The ***Wetted Surface*** is a key factor when estimating the drag of the hull.

The ***Top Area / Center of Buoyancy*** factor can be used for estimating the trim balance of a planning hull. A good trim is usually achieved by placing the top area center a few percents of the waterline length ahead of the center of buoyancy.



Offsets.

The *Offsets* command generates a *3D coordinate table* of the selected object. The table is used for creating full-scale cross sections of the surface, which can be placed in a building jig. The jig is used for holding the skin elements while building the object. The table can be copied to the clipboard, be saved as a file or be printed.



		0	500	1000	1500	2000	2500	3000	3500	4000	
	H400	x	x	x	x	x	594	538	377	x	H400
1	H200	x	x	416	544	562	525	434	268	x	H200
	H0	x	x	x	x	x	x	x	x	x	H0
	EHX400	-2039								1927	
	EHZ400	0								0	
2	EHX200	x								1884	
	EHZ200	x								0	
	EHX0	x								x	
	EHZ0	x								x	
	V600	x	x	x	x	194	212	x	x	x	V600
	V400	x	127	-6	-84	-111	-101	-37	238	x	V400
3	V200	196	68	-51	-123	-148	-148	-125	-54	x	V200
	V0	190	59	-66	-136	-155	-157	-141	-99	x	V0
	EVX600	x								x	
	EVY600	x								x	
	EVX400	x								x	
	EVY400	x								x	
4	EVX200	x								x	
	EVY200	x								x	
	EVX0	-1585								1850	
	EVY0	79								-20	
	RulY0	x	59	-66	-136	-155	-157	-141	-99	x	RulY0
	RulZ0	x	0	0	0	0	0	0	0	x	RulZ0
	RulY1	x	112	92	73	53	33	13	-6	x	RulY1
	RulZ1	x	0	0	0	0	0	0	0	x	RulZ1
5	RulY2	x	153	138	123	108	93	79	64	x	RulY2
	RulZ2	x	0	0	0	0	0	0	0	x	RulZ2
	RulY3	208	204	200	195	191	187	182	178	x	RulY3
	RulZ3	0	0	0	0	0	0	0	0	x	RulZ3
	Corn1X	-1585									
	Corn1Y	79									
	Corn1Z	0									
	Corn2X	-2073									
	Corn2Y	209									
	Corn2Z	0									
6	Corn3X	1927									
	Corn3Y	302									
	Corn3Z	0									
	Corn4X	1850									
	Corn4Y	-20									
	Corn4Z	0									

The ***first (1)*** and ***third (3) sections*** contain the horizontal and vertical coordinates as defined in the Sectioning setting dialog (Page menu).

The ***second (2)*** and ***forth (4) sections*** define where the horizontal and vertical sectioning cuts meet the edge of the surface. Note that these coordinates come in pairs.

The ***fifth (5) section*** defines where the control rulers meet the surface. Note that these coordinates come in pairs.

The ***sixth (6) section*** defines the four X, Y and Z corner coordinates.

Extrude, and other drawing commands in the Tool menu

The ***Extrude, Line, Rectangle, Circle, Polygon, Double Line, Arc, Surface, Point, Cube, Sphere, Torus, Cylinder, Circle to Rectangle Extrusion and T-tube*** commands are all essentially the same as found in the Tool palette. We therefore recommend that you take a look in the ***Tool palette section*** of this manual to find out more about the respective features of these tools.

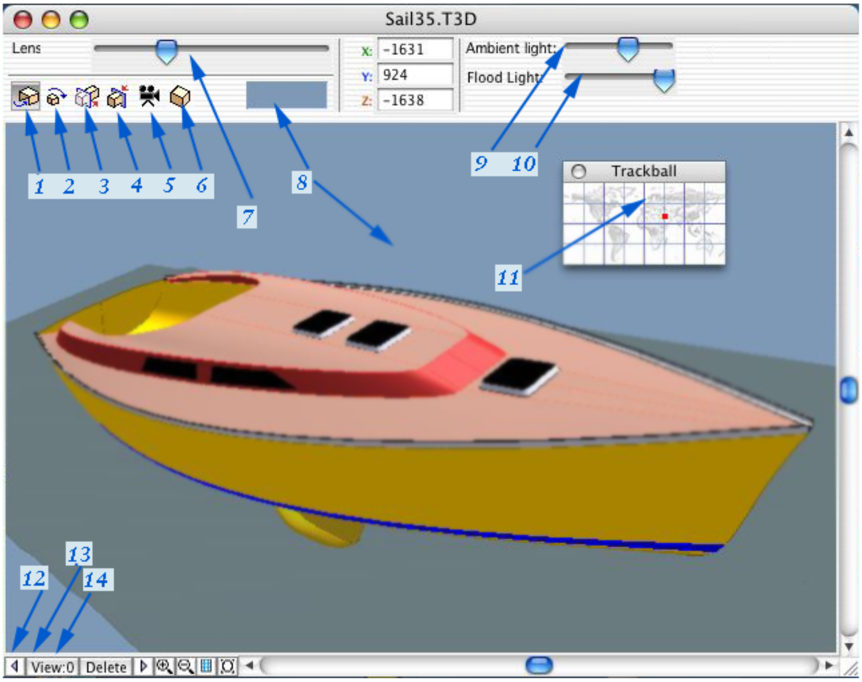
The difference between the tool commands in the Tool menu and the tools found in the Tool palette is that they allow you to ***generate an object*** without having to select a tool in the Tool palette and draw something in the drawing area first. It can sometimes be practical and may save some time. Defining a shape using a dialog can however be a bit abstract, but can sometimes be quicker and more accurate.

An ***alternative way*** of drawing an object by means of a dialog is to ***double-click*** on the relevant ***tool*** in the ***Tool palette***. This method allows you to skip the drawing part and go directly to the object creation dialog.

The ***Extrude*** command has a ***quick-key [command + E (M) / control + E (W)]***

View menu

The *View* menu contains various viewing settings such as rendering, views, and what you see in the model editor.



Render model

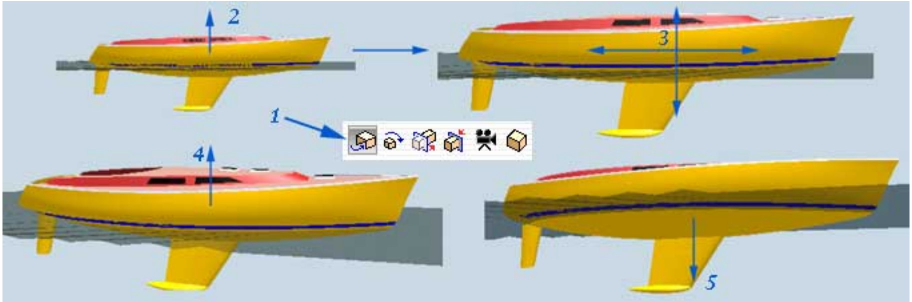
A model is normally displayed as a wireframe model while editing TouchCAD allows you to see a rendered version of the model. All visible surface objects can be rendered.

The *color* of a *surface* is controlled by the *fill color setting* and can be changed using the *color* pop-up in the *Object Info* palette.

A *color is a group of color settings* consisting of a *name*, a *line color*, a *fill color* and a *transparency* setting. Colors can be added and deleted using the *Color* tab of the *Layer/Color* palette.

Choose *Render Model* in the *View* menu. The model is now displayed as a solid surface. Some new controls are shown in the drawing window when you render the model. We will now take a closer look at the options.

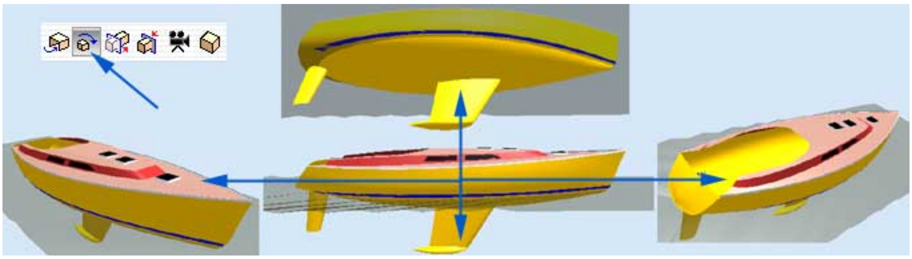
The controls are *Walkthrough (1), Flyover (2), Hither (3), Yon (4), Animate (5), Render / WireFrame (7). Background Color (8), Ambient Light (9), Flood Light (10), Trackball palette (11), Step backward view (12) Save View (13), and Step Forward view (14).*



Walkthrough tool

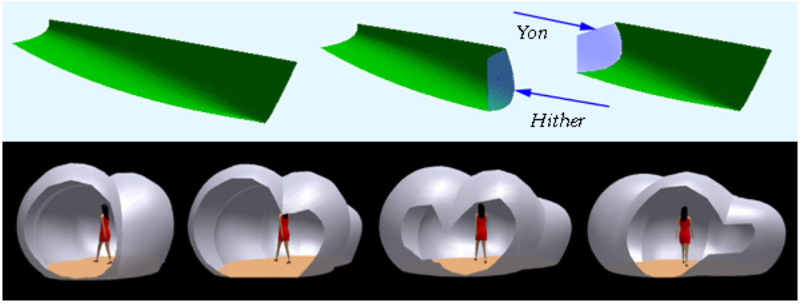
The ***Walkthrough (1)*** tool enables you to walk through the model. You operate it by placing the ***cursor*** close to the ***center*** of the rendered area (2). ***Press the mouse button*** and ***drag*** the cursor ***upward*** to go ***forward***. The model moves closer to you ***(3)***. Drag ***downwards*** and you move ***away*** from the model. ***Drag left*** and you rotate to the ***left***, ***drag right*** and you rotate to the ***right (3)***. Press the ***Alt / Opt key***while dragging ***upward*** on the screen moves the camera ***upwards (4)***, ***down*** for ***downwards (5)*** and ***left/right*** for moving ***sideways***.

Note that the ***numerical view keys*** also work. The only difference is here that the ***zero key*** is used for ***resetting the view*** instead of unfolding the model.



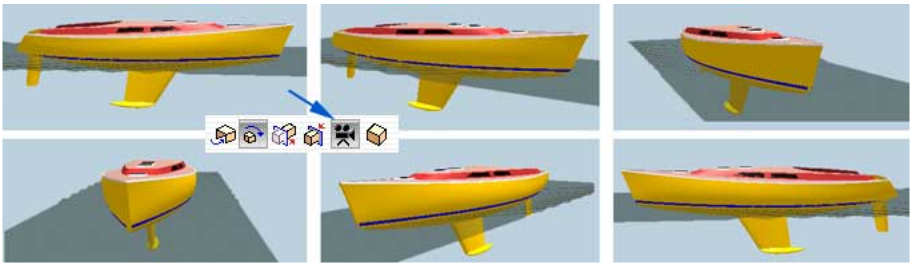
Flyover tool

The ***Flyover tool*** enables you to rotate the view in the same way as when using the ***Trackball***. Hold the cursor over the drawing area and drag to rotate. Unlike the Trackball the ***Flyover*** tool also allows ***recording of a simple animation***. More on that later on.



Hither and Yon tool.

The *Hither* and *Yon* tools are used for *cutting off parts* of the model *from* the *front* and *back* sides. This enables you to look *into a section of the model*. It also works when you record and play an animation, which can generate certain useful effects such as shown in the lower section of the picture above. *Dragging upwards/downwards on the screen operates both tools*. Note that the *numerical settings* are displayed in the *Info* field above the drawing area when you drag.



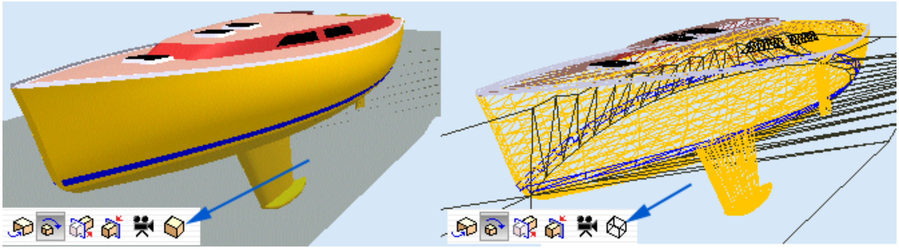
Animate tool

It can sometimes be useful to see a model in motion when designing certain objects. The *Animate tool* enables you to *record a motion* and *play* it in *an endless loop*.

Choose the *Front* view by pressing the *2 key* on the numerical keyboard. Choose the *Flyover* tool and *rotate the view slightly*. Click on the *Animate* tool and the model starts *rotating*. Click again on the *Animate* tool to stop the motion.

Note that you can *lock the motion* while *dragging / recording* the motion to being either *constrained horizontally or vertically* by pressing the *Shift key*.

You can change view at any time while running the animation, useful if you only want to see a small part of the entire motion loop.



Hidden Lines /Solid display tool

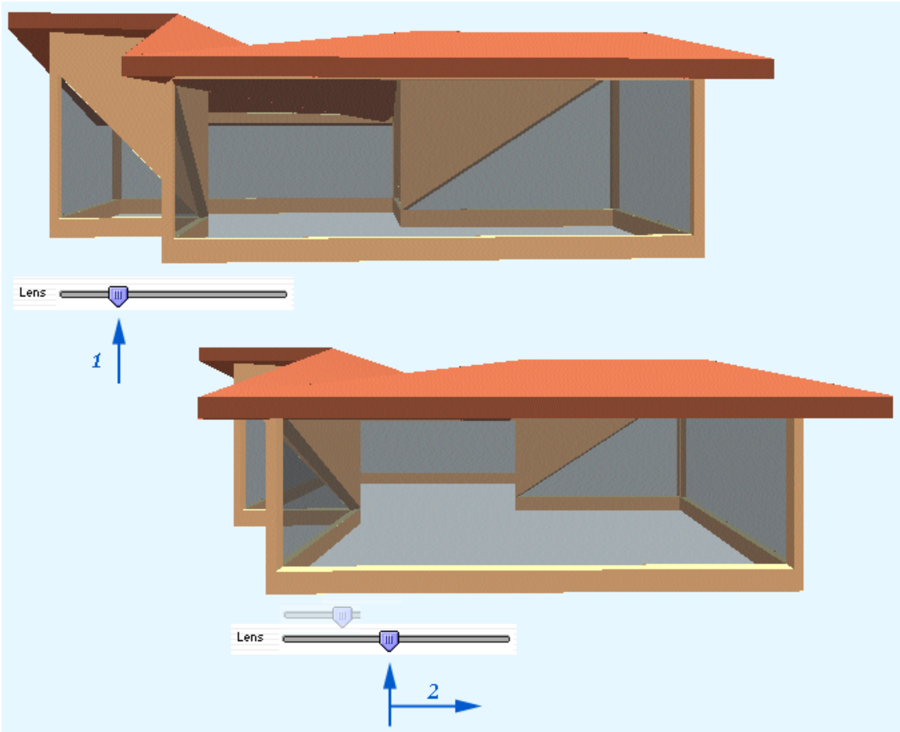
This tool enables you to toggle between looking at the model as a wireframe or as a solid.

Background color tool

The ***Background*** field enables you to set the color of the background by choosing any color. Just click in the color field and chose a color using a standard color-dialog.

Light settings

These two ***Light sliders*** allow you to dynamically change the ***Ambient*** and ***Flood*** light settings. The ***Ambient light*** control makes the rendering look ***lighter or darker***. The ***Flood light*** control ***increases*** and ***decreases*** the ***contrast***.



Lens Setting tool

The ***Lens*** setting tool allows you to set the *perspective deformation* of the display.

Some rendering notes

The ***Skin type*** setting in the ***Object Info*** palette also affects the way a surface is rendered. Smoothing in a certain direction generates a smooth rendering in this direction.

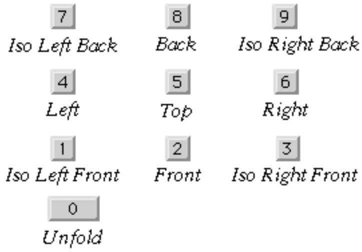
Views saved with the ***Save*** button in the bottom left corner of the drawing window are also saved when the document is saved. The saved view includes ***view-, perspective-, lens-, light-, yon-, hither and background color*** settings.

There are ***separate lists of saved views*** for the ***WireFrame*** and ***Render*** modes, which can store up to ten views each.

A good suggestion is to check out the various display modes found in the Properties tab of the Object Info palette. The Display pop-up enables various settings such as wireframe, outline, mesh, filled mesh, visible panel joints, or various combinations thereof to be displayed in the rendering mode. A more detailed description can be found in the Object Info palette section, Property Tab.

Standard views -> ....

The ***Standard views*** are a series of fixed views. You will find some of these views in the View bar over the drawing window. They are also available on the numerical keyboard according to the following layout.

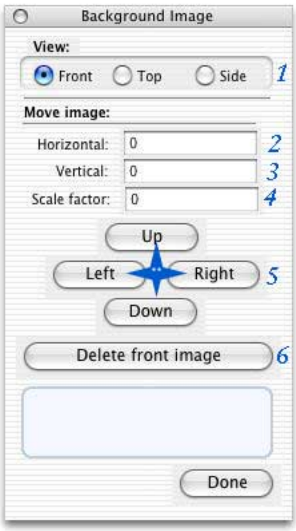


Show Background Image

The Show Background Image commands enable background images to be made visible or not. Visible images have a check mark.

Edit Background Image...

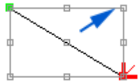
The Edit Background Image command displays a floating palette that enables fine-tuning of the location of the background images relative to the 3D model and in scale.



The first three ***View buttons (1)*** state the ***target view*** image to be edited. The ***Horizontal** (2)* and ***Vertical** (3)* edit fields move the image by value. The ***Scale factor** (4)* edit field provides a method to scale a image relative to the 3D model by a scale factor in percent. The Nudge buttons (5) moves the image one grid step relative to the 3D model. The ***Delete** (6) **button*** deletes the currently selected image from the file.

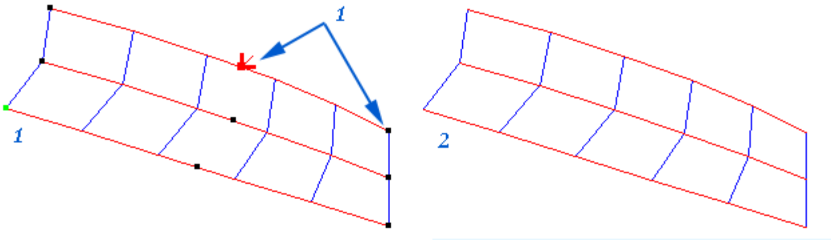
Low Res. Dragging

The *Low Resolution* dragging option is used when you rotate and move complex objects. In such a case it can be practical to ***reduce*** the ***mesh resolution while dragging***. This is done automatically if this option is checked. Note that it does not have any effect on the model when it is static.



Show Resize Box

The *Show Resize Box* command allows you to turn on or off the Resize box feature in the drawing area. The ***quick-key*** command ***is [command + < (M) / Control + < (W)]***. Please refer to the ***Resize Box*** section found in the ***Basic Concept*** chapter for further details.



Rotation box

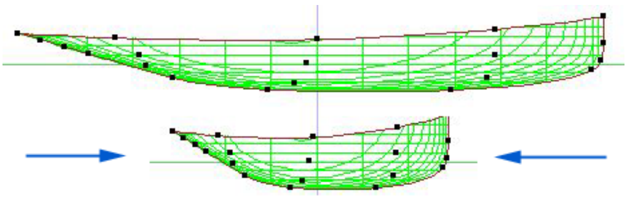
The Rotation Box option enables you to turn on and off the rotation features of the Resize Box. Please refer to the ***Resize Box*** section found in the ***Basic Concept*** chapter for further details.

Show Controls

The *Show Controls* command allows you to ***turn on (1) and off (2) the control dots***. This is primarily used when presenting a final model.

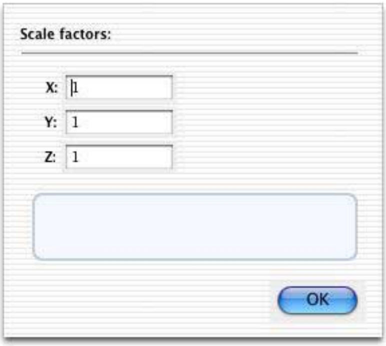
Coordinate axes

The Coordinate Axes command allows you to turn on and off the coordinate axes found in the drawing window.



Analyze Curves

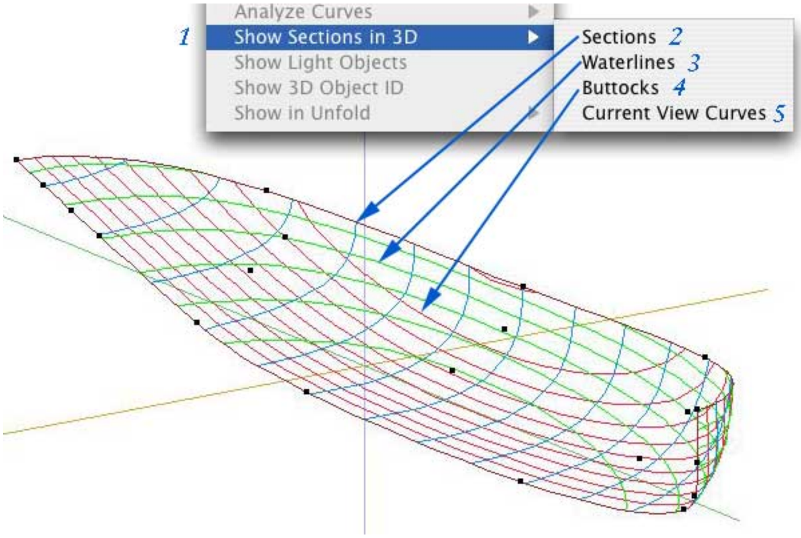
The *Analyze Curves* commands enables you to apply a ***temporary stretch factor*** on one or several of the basic X-, Y-, and Z- axes. It is primarily used for analyzing the lines / curves of a shape to locate areas of bad fairing.



The ***Stretch-X, Y and Z*** items simply turn on the stretching in the given direction.

The ***Stretch Settings…*** dialog specifies the degree of stretching. A factor of 1 means no stretching 0.5 scales the direction to half the original size and so on.

You will find ***another shape analyzing method*** when activating ***the Surface Analyzer*** palette found in the Palettes menu.



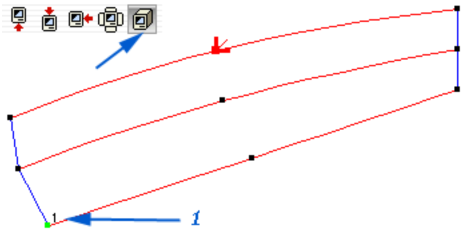
Show Sections in 3D

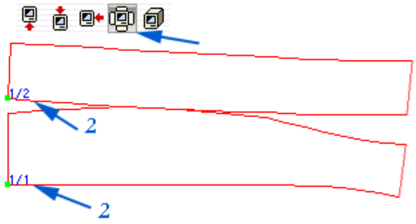
The Show Sections in 3D (1) commands control how the dynamic cross sectioning features is displayed. The ***Sections*** *(2)* option shows cuts along the X-axis. The ***Waterlines*** *(3)* option shows cuts along the Y-axis. The ***Buttocks*** *(4)* option shows cuts along the Z-axis. These features are on by default but it may be easier to read and it is certainly faster to use a more limited setting from time to time. ***The Current View Curves*** *(5)* just displays the relevant curves in the respective Front, Top and Side views, which makes it a bit quicker as it does not need to calculate curves that are straight anyway.

The ***sectioning distances*** and other sectioning settings are controlled by the ***Sectioning Settings…*** command found in the ***Page*** menu.

Show Lights

The *Show Lights* command makes light objects visible or not. Note though that it only displays light objects that are turned on. You handle the light properties in the ***Lights palette*** found in the ***Palette*** menu. Lights only affect the rendering mode.





Show Object ID

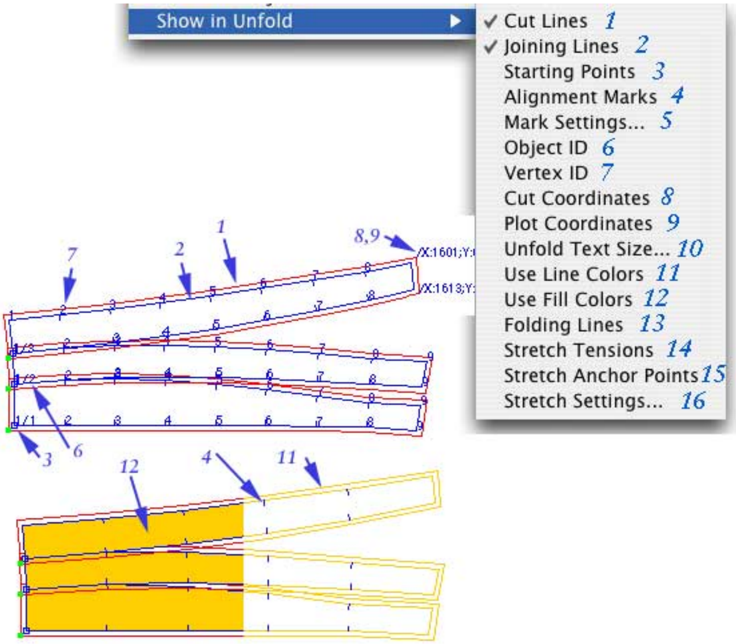
The *Show Object ID (1)* command allows you to see the ***object ID*** in ***3D-views*** close to the ***green starting point dot***. The main use for this feature is to provide a reference picture when assembling unfolded objects.

The ***unfolded equivalent*** is found in the ***View menu*** *->* ***Show In Unfold*** *->* ***Object ID*** *(2)*. This feature places an ***ID text*** close to each ***sub-panel starting point***. The ***syntax is 1/1, 1/2, etc.*** where the ***first number*** specifies the ***main panel ID*** and the ***second the sub-panel ID***. Note that the sub-panels are always located in consecutive order starting from the main panel starting point, and this makes it easier to assemble the parts in a correct way.

On more complex models we recommend you to generate a 3D assembly-plan that can be used as a map when building the model.

Show in Unfold

TouchCAD provides a range of ***general unfolding settings***. These options can be found in the ***View menu -> Show in Unfold -> etc.***



These options allow you to organize and keep track of very large and complex models consisting of hundreds of sub-panels. All these unfold options are supported when exporting as DXF (supported by most CAD programs), VectorWorks (VectorScript) and PlotMaker (PlotMaker is a program for controlling sail panel cutting machines used by many sail makers).

1/ The ***Cut Lines*** option shows the outer edge of the surface including overlaps.

2/ The ***Joining Lines***, that is where the final surface edges meet excluding overlaps.

3/ The ***Start Points*** are used for specifying where each panel starts. The Starting point of a main panel is the green dot and it is visible in 3D too. Note that there is a Show Object ID higher up in the menu and that option allows you to see the Object ID number in 3D too.

4/ Alignment Marks (sometimes also called strike up marks) plot lines along the edge of the panel and help to align surface edges when assembling panels.

5/ The ***Mark Settings*** specify the length of the alignment marks.

6/ Unfold ***Object ID*** inserts a ***panel number*** in the ***starting point corner*** in the following format (5/3). The first number specifies the main panel number and the second is the sub panel number (e.g. 5/3 means that the main panel number is five and it is the third sub panel in that main panel). Note that the sub-panels always occur in consecutive order with the starting point in the same corner. This helps to keep track of where they belong.

7/ The ***Vertex ID*** plots the vertex number along each side. This helps when assembling large panels with many points along the edge.

8/ The ***Cut Coordinates*** option plots the ***coordinates*** along ***the true edge of the panel.*** This can be useful when optimizing the use of material to fit into a given material size. Note that each panel has its own local coordinate system, and it will dynamically update as you move and rotate the object. It even allows you to edit the 3D-model with the arrow keys while being in the Unfold view. Just select the point in 3D and then switch to the Unfold view and use the arrow/+/- keys as if you wherein the Front view. This works because the 3D model is dynamically linked to the unfolded model, so any changes in 3D can instantly be seen in the Unfold view. If you look closely you will see that the unfolded shapes and coordinates slowly changes as you move. Optimizing the use of material can therefore be an integrated part of the design process in an efficient and powerful way.

9/ The ***Plot Coordinates*** are basically the same as the Cut coordinates but they represent the ***inner joining line*** instead.

10/ Unfold Text Size sets the size of the text in the Unfold view.

11/ Use Line Colors displays the 3D-line color instead of the standard unfold colors.

12/ Use Fill Colors display the 3D fill color instead of a white outline. It can be used for printing out color scale models on a color printer.

13/ Folding Lines displays lines across the unfolded panels, which can be useful when fabricating sheet metal objects using a folding or rolling machine.

14/ Stretch Tensions command makes the ***tension markers visible***. More ***on Stretch Unfold features*** in the ***Object Info -> Unfold Tab*** palette section of the manual.

15/ Stretch Anchor Point command makes the ***Anchor point marker visible***. More ***on Stretch Unfold features*** in the ***Object Info -> Unfold Tab*** palette section of the manual.

16/ Stretch Settings... dialog defines how the ***stretch tension markers*** are being displayed. More ***on Stretch Unfold features*** in the ***Object Info -> Unfold Tab*** palette section of the manual.

Unfold export limitations.

All these options can be printed and exported to other programs. There are however some limitations:

The DXF export does not support fills.

The PlotMaker export does not support fill colors and coordinates.

The HPGL export does not support fill colors and coordinates.

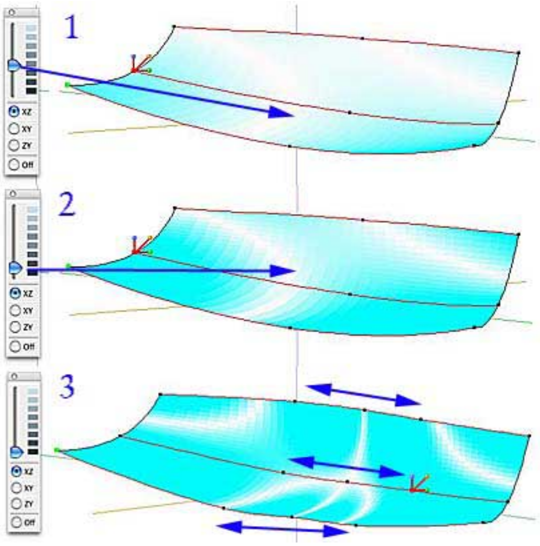
The VectorWorks export supports all options.

The Unfolded image export supports all options.

Palette, About TouchCAD menu

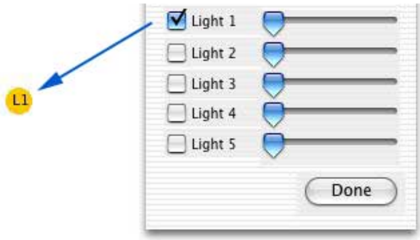
Palette menu

The ***Palette menu*** is used for showing and hiding the five standard palettes used by TouchCAD. Most of the features of these palettes are covered in other sections of the manual though two ore not.



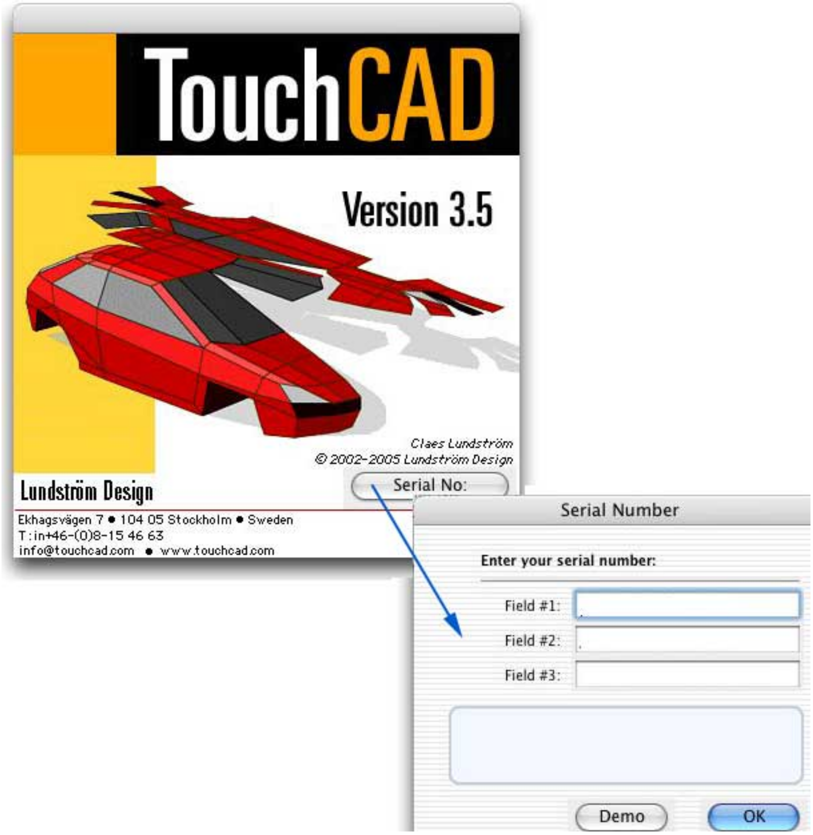
Surface Smoothness Analyzer.

The *Analyze Smoothness palette* adds a new set of tools for graphically ***analyzing the smoothness*** of a surface in the X-Y, X-Z and Z-Y directions. Rapid changes in the change are displayed as highlighted areas within a given selected surface *(1-2)*. Note that highlights defined as smooth transformations may not necessarily be a bad thing, but rapid changes up and down in color may suggest a not so smooth shape *(3)*. The analyzing process is dynamically updated as the shape is updated, when you change direction or when you change the setting of the sensitivity slider.



Lights palette

The *Lights palette* controls the light settings when rendering the model. A basic light is automatically inserted so these lights are basically used for adding and highlighting certain areas. Light objects become visible in the drawing window if they are checked in the palette and can be dragged to a suitable location as any other 3D object. Note that the Show Lights item in the View menu must be checked to make them visible.



About TouchCAD

The ***About TouchCAD*** menu item can be found under the Apple menu on MacOS and under the Help menu on Windows.

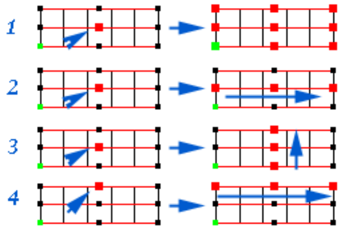
This command displays the ***About TouchCAD*** dialog. Here you will find a button that takes you to the ***Serial number*** dialog.

Contextual menu

TouchCAD displays a contextual menu when control-clicking (Macs with a one mouse button) or right-clicking (on Windows and Macs having a two or more button mouse installed) in the drawing window. The options are:

Select All, which selects all accessible objects.

Deselect All, which deselects everything.



Selects All Points in the Object *(1)*, which selects all control points in the object having the Intersection point.

Selects All Points Along the Ruler *(2)*, which selects all control points along the control ruler object having the Intersection point.

Selects All Points Perpendicular to the Ruler *(3)*, which selects all control points having the same point index as the control point having the Intersection point.

Selects All Points Along the Edge *(4)*, which selects all control along the surface edge the where the Intersection point is located. Note that you need to select a point that explicitly belongs to one side to make this work. A corner point will not work.

Undo, which steps back one step in the action list.

Redo, which steps forward one step in the action list, if you have used the Undo command before.

All these commands can also be found in the standard menus and can also be reached using keyboard shortcuts.

Keyboard shortcuts

TouchCAD provides many keyboard shortcuts for various tools and commands. Some of them are used in combination with the Command (Mac) / Control (Windows) keys to work, whereas others activate something using a single key.

Command key shortcuts

Below is a list of the commands requiring the Command / Control keys (here represented by the # character) being pressed to work:

File menu

New #+N
Open #+O
Save #+S
Print #+P
Quit #+Q

Edit menu

Undo #+Z
Redo #+Y
Cut #+X
Copy #+C
Paste #+V
Group #+G
Ungroup #+U
Ignore Groups #+Shift+G
Move #+M
Reshape #+R
Rotate Left 90 ° #+L
Rotate… #+Shift+V
Select All #+A
Select All Points in Object #+Shift+6
Select All Point Along Ruler #+5
Select All points Perpendicular to ruler #+Shift+5
Select All Points Along Edge #+6
Duplicate #+D
Duplicate Array #+Shift+D
Control Pt Resolution #+Shift+A
Redistribute Controls #+Shift+J
Join Objects #+J

Page menu

Fit to Window #+4
Set Grid Resolution Bigger Steps #+2
Set Grid Resolution Smaller Steps #+1
Set Origin #+9
Reset Origin #+0

Tool menu

Marine Properties #+K
Extrude #+E

View menu

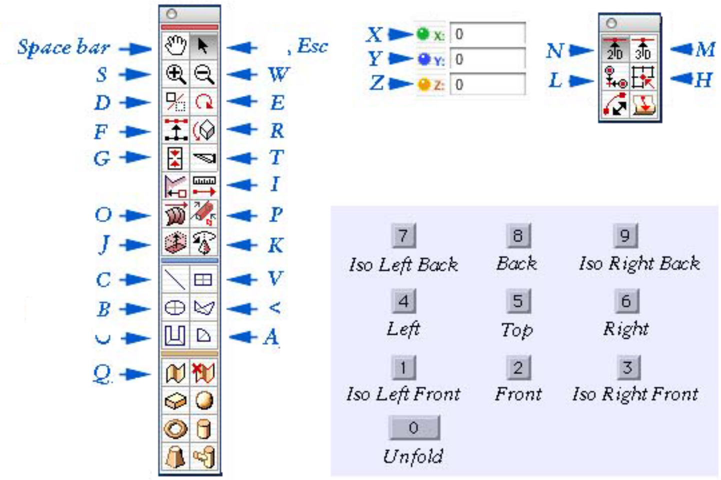
Render Model #+H
Show Resize Box #+<

Palette menu

Object Info #+I
Tool #+T

Single key shortcuts

Below is a list of keyboard shortcuts where you press a single key to activate a tool or command.



Note that the Escape key can be used for canceling a tool action, and when such an action takes place the tool is set to the default tool, which is the Selection Arrow. If no action has started, the Esc key takes you to the Arrow key.

The Space bar temporarily deactivates the tool used and switches to the Pan tool. Note that you do not need to press the Space bar while panning, just press the Space bar again to go back to the tool originally used.

Nudging.

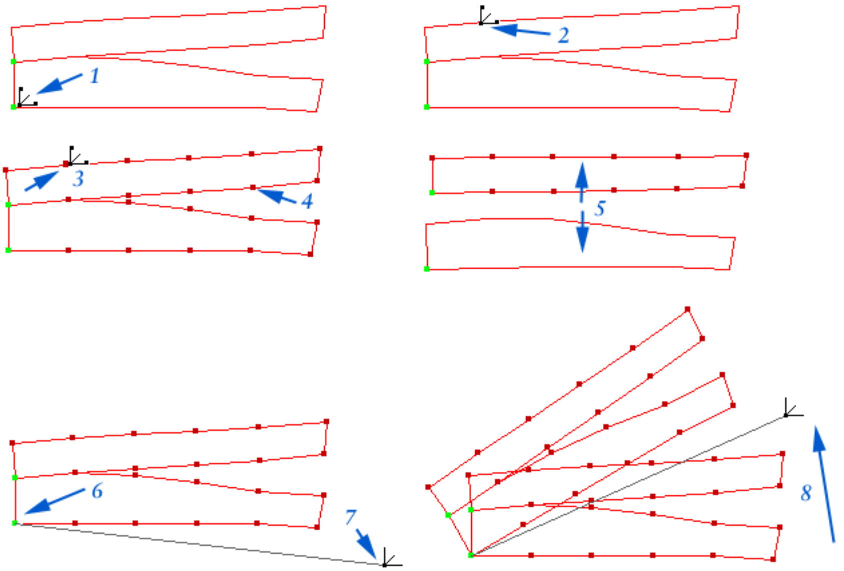
The ***arrow keys*** are normally used for ***moving selected objects*** on the screen but they can also be used for other things. Press ***Shift + an arrow key*** to quick ***pan***. Press ***Alt/Opt + an arrow key*** to **nudge-*rotate the 3D view. Press Command (Mac)/Windows flag (Windows) + an arrow key*** to ***nudge zoom in and out.***

Trackball keyboard features.

The Trackball palette also provides some keyboard-controlled features. Press ***Control plus Click*** in the window to display the ***Trackball Sensitivity*** slider. Click again to deactivate it. ***Press Opt/Alt while dragging*** with the mouse within the Trackball window to ***zoom in and out. Press Shift while dragging*** with the mouse within the Trackball window to ***pan***.

Working in the Unfold view

The unfolding feature in TouchCAD is essentially an ***unfolded 2D-projection*** of the 3D-model, that is, a ***view having*** a ***dynamic link to the 3D model***. This link means that any ***changes*** in the ***3D*** model can instantly be seen in the Unfold view and are ***instantly available*** when the ***Unfold view is selected***.



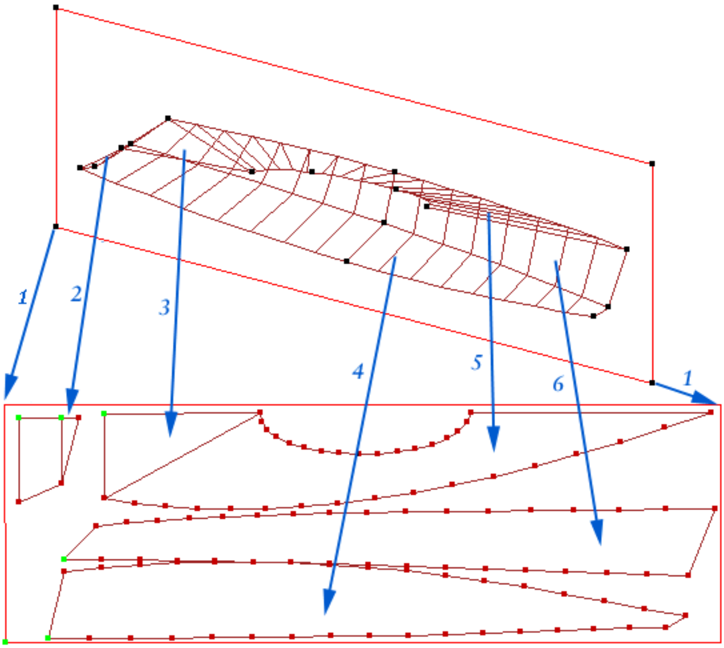
Using the Unfold view

Objects can not be drawn in the Unfold view but you can ***edit*** many of the ***properties*** such as ***skin resolution*** and ***unfolding direction***. You can also ***move objects*** by ***dragging or nudging*** and ***rotating*** them. The ***Selection Arrow*** is used for selecting the object you want to edit. ***Clicking*** on the ***green starting point (1)*** of an object ***activates*** it, and ***red dots (4)*** along the edge indicates that it has been selected. An object can also be ***activated*** if the ***cursor*** (2) ***indicates*** that you have a ***snap***, that is, it displays red dots at the end of the lines.

Any of the red curve dots around the object can be used as a ***handle (4)*** for dragging the object.

Objects can also be ***rotated*** using the ***Rotate*** tool in the Tool palette. The method works in exactly the same way as in 3D, though you can not use the Constrain rotation modes. Select the object to be rotated, click (6) in the center of rotation, drag the rubber-band line to the second location and click (7) to establish the lever arm, drag to rotate (8) and then click when the angle is where desired.

Sub-panels can not be moved individually, but you can convert the ***elements*** of a ***main panel*** into ***individual*** objects using the ***Split*** (5) commands found in the ***Edit*** menu.



Creating a cut layout

A useful technique to optimize the use of material is to create a ***cutting layout***. A recommended method is to draw a ***rectangular test surface*** used for representing the available sheet dimensions. This makes it easier to optimize the use of material. You simply ***drag in*** the unfolded elements ***into the test rectangle*** and check if they fit in. Above is a simple example.

The ***red (1) rectangle*** represents the ***sheet used***. It can be placed in a separate layer if you want the ability to make it visible or invisible. The rest of the ***objects (2-6)*** are the ***actual unfolded pieces***. This makes the layout very clear and it can be made very efficient if you experiment a bit.

Using the Arrow keys in the Unfold view

One way of ***optimizing*** the use of material would be to make ***small changes*** in the ***3D-model*** and then click on the ***Unfold view*** to see if the ***result*** is acceptable or if it needs some more nudging. This is perfectly OK due to the speed of the unfolding calculations but there are certain useful features allowing you to ***edit*** the ***model*** without changing views a lot.

Three buttons *(2-4)* occur in the ***Mode*** bar when you select the ***Unfold*** *(1)* view. These three buttons controls the behavior of the selected objects when nudging with the arrow keys.

Unfold nudging

The ***left button*** *(2)* is used for ordinary ***nudging / moving*** the ***unfolded objects*** in the ***Unfold view***, that is, change its physical location within the Unfold view.

3D nudging

The ***second button*** *(3)* ***moves selected 3D-control points in the 3D views while being in the Unfold view.*** In this mode, ***the arrow and plus / minus keys behave as if you were in the Front view.*** The Left and Right arrow keys increase and decrease the X coordinates one grid step. The Up and Down arrow keys increase and decrease the Y coordinates one grid steps. The Plus and Minus keys increase and decrease the Z coordinates one grid steps.

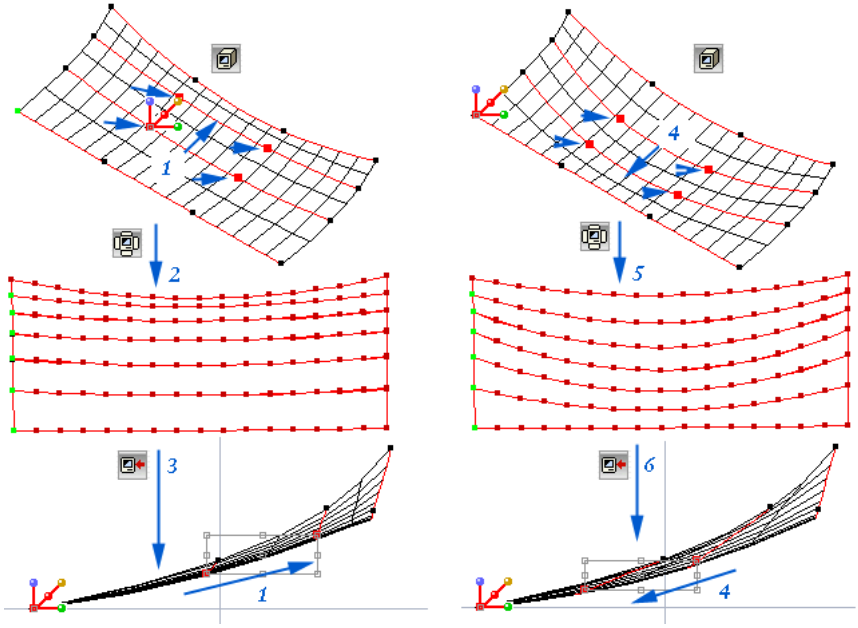
An example: Imagine that you have designed an object, and intend to use sheets of material 2000 mm wide. When you unfold the model it turns out that the model has an unfolded width of 2007 mm, which is close but still too wide. Rotating the model to the optimal angle using the Rotate tool in the tool palette doesn't help. The solution is to select one or more points in the 3D-model, select the Unfold view and nudge the selected point until it fits into the 2000-mm sheet width.

Note that the ***Set /Reset Origin*** commands can be useful for such jobs. Displaying the ***cutting coordinates*** (***View -> Show in Unfold -> Cut Coordinates***) while optimizing the use of material is another useful feature.

Projected 3D unfold nudging

The third button in the *Mode* bar works in a similar way to the 3D Unfold Nudging. The purpose of this mode is primarily to allow ***nudging of control points*** being located on the surface ***along the actual surface*** of the model.

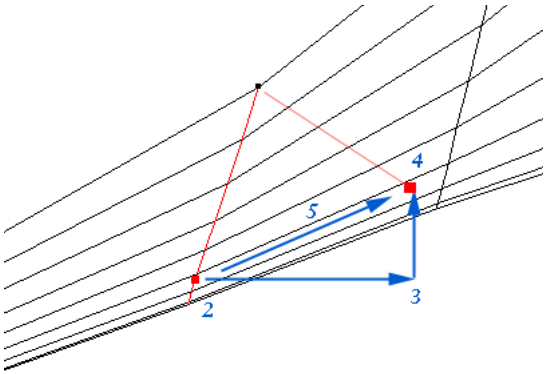
The advantage with this method is that it allows you ***to change the distance between the rulers without changing the shape*** of the surface too much. This change of distance between the rulers ***affects the width of the strips***, and allows you to "***borrow***" material (widths) from areas where the strips are narrower than the maximum material width and ***transfer*** it to where the ***strips are too wide to make them narrower***. Below is an example.



The four controls in the ***middle*** *(1)* have been selected, and have been nudged ***upwards*** in the ***Unfold*** *(2)* view. The sliding takes place ***along the surface***, like water running along the skin (if you can imagine water running upwards).

The same steps but in the ***opposite direction*** is illustrated in the ***steps*** *4-6*. Note that the ***shape*** *(3 & 6)* is more or less ***the same*** if you compare the two side views, but the ***unfolded panel layout*** *(2 & 5)* is very different.

The example illustrated below shows that you could improve the use of material a lot. The practical use may still be a bit abstract and for that reason we will take a closer look at the steps and explain what happens.



The picture shows a ***side view*** of the model. The following steps illustrate how to think and what to do.

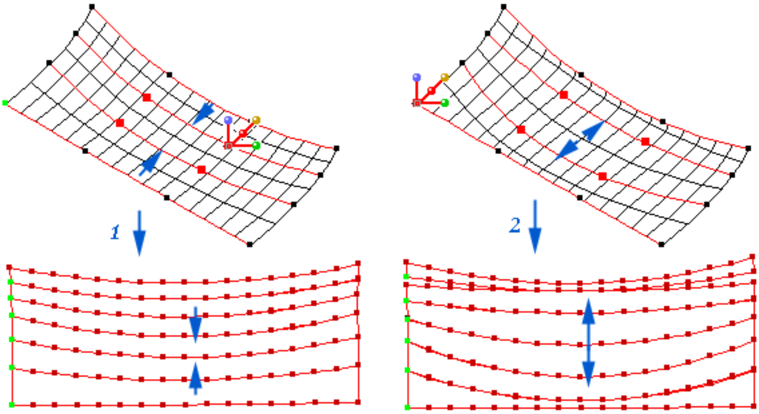
A/ The model is a sail roof and has the ***biggest extension*** seen from the ***Top view***, and seen from that view the rulers have a horizontal orientation. This is a key factor, and means that we will ***slide the controls in Z direction***. The ***Arrow keys behave*** as if you were in the ***Front view*** when you use this ***mode*** *(6)*. In ***the Front view*** you ***nudge objects in the Z direction*** using the ***Plus- and Minus- keys***, and these are the keys to use in this case.

B/ Select the ***3D-points in a 3D view*** and then ***select*** the ***Unfold*** view. Select the ***Slide mode*** *(6)* and the ***Y projection*** *(7)*.

C/ This is what happens when you use the ***Plus- and Minus- keys***. Note that we show what happens when seen from the side but the action takes place in the Unfold view. If we press the ***Plus key***, the ***3D Z coordinate*** is ***increased*** by ***one grid step*** and we end up at the ***third*** *(3)* location. This would change the shape of the skin, and that is not what we want, so before the shape is updated, ***TouchCAD projects the control point*** to the ***skin as it was before we pressed the Plus key***. This moves the control point ***upward*** to ***position five*** *(5)* because we used the ***Y projection method*** *(7)*. The skin is then recalculated. The ***practical result*** is that the ***rulers slide*** as illustrated by the ***diagonal*** *(5)* ***vector arrow***, in other words ***along the surface skin***.

Intermediate rulers

The intermediate rules can also be used for redistributing the unfolded strips over the skin. Below is an example.

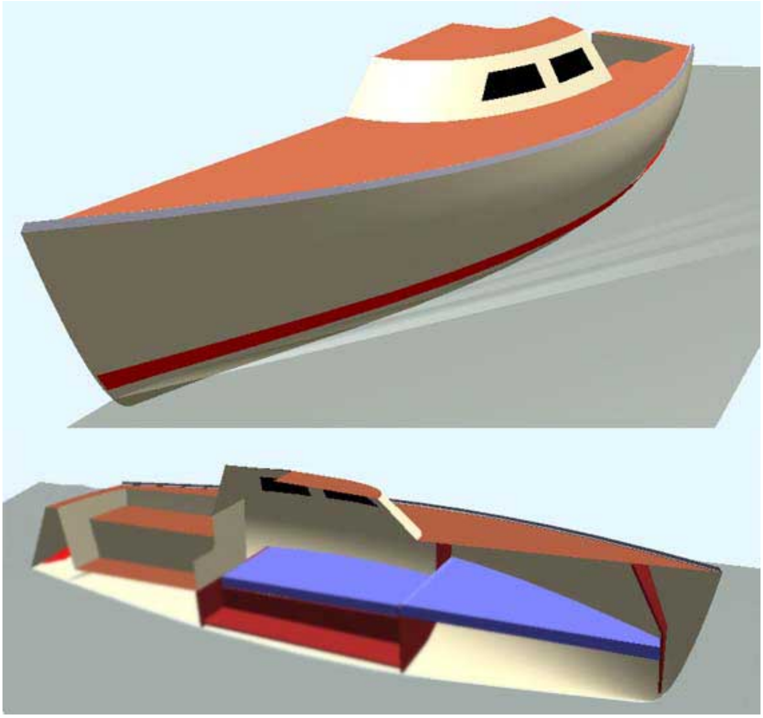


In the ***first*** example the ***control rulers*** have been pushed closer to the ***center*** of the ***surface***. In the ***second*** example they have been moved ***closer*** to the ***edge*** of the ***surface***. Note the ***difference*** in ***panel width***. It is obvious that it can be useful to take some time to ***optimize*** the ***use of material*** in this way, and this is how ***the 3D sliding method*** comes in. It allows you to move the curves without worrying too much about the 3D shape, at least if the changes are reasonably small.

Some suggestions

The above described technique should be used with moderation though. ***Extreme changes*** may cause ***deformations*** in the shape. The ***Change Grid Resolution*** commands found in the ***Page*** menu are quite useful features while ***nudging*** on large models. It offers a quick method for controlling the speed of the motion.

Tutorial, a boat hull

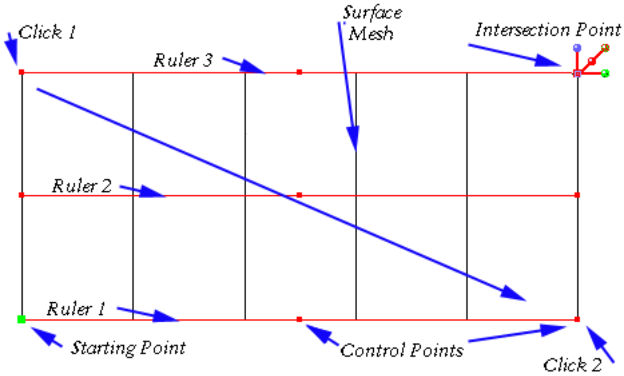


The object of this exercise is to draw a boat hull and some other components found in a boat. This illustrates many common techniques used when modeling with TouchCAD, such as extruding connecting surfaces and projecting control points on other surfaces.

Drawing a surface

Start by drawing a surface. This will help you understanding some basic features of the program. Choose the ***Front*** view by clicking on the left button in the ***View bar***. Click on the *3D-Surface* tool in the *Tool palette*. Move the cursor to the drawing surface and draw a rectangle. Note that you don't have to press the mouse button while dragging. Just click at the starting and end points. Ignore the dialog that shows up for now. Just click on the OK button. The picture below shows the drawn surface.

Front view *3D-Surface tool*



Surface objects

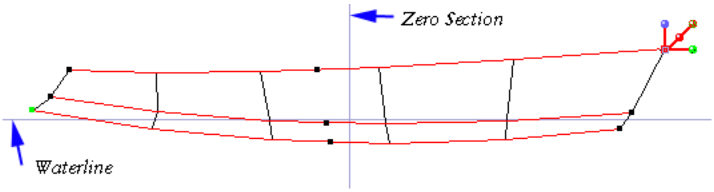
The ***green dot*** represents the ***starting point***. The ***red dots*** are ***control points*** used for changing the shape of the surface. In this case they are all ***red***, meaning that they all are ***selected***. ***Unselected*** control points are ***invisible***. The ***green starting point*** is also a ***control point***..

Dragging the Surface

Choose the *Selection Arrow* from the *Tool* palette (the top right item). Click on one of the red dots. The surface now follows the movements of the cursor. Note that you don't need to press the mouse button while dragging. Click again to stop moving the surface.

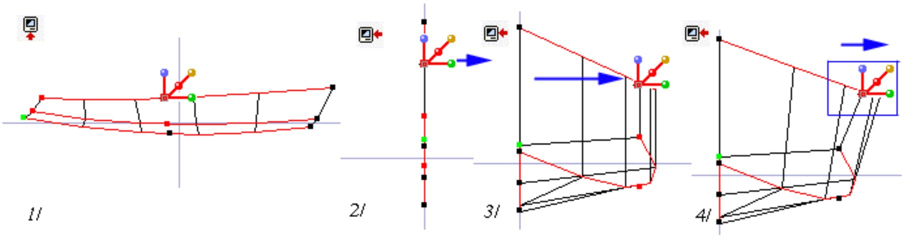
Editing the surface

The entire surface was moved because all control points where selected. In order to just move one control point; click beside the surface in the drawing area. All control points become black to indicate that they all are deselected. Click on one of the points. The red Intersection point indicator appears again indicating that it is selected. Click again to start the moving sequence. Click again at the new location. Note that the surface changes shape when you drag. Try moving some other control points. Note that you can Shift-Click to activate further points. You can also draw a selection rectangle, where every control point inside will be selected.



Adjusting the shape

Now, try modifying the surface so that it looks roughly like the picture above. Try to align the surface with the zero section and the waterline. They are displayed as light blue lines. If you click on the ***Right/Side view button*** in the ***View bar*** you will only see a vertical line. This means that the surface is completely flat (**2**). We will now add a third dimension to the model.



Click on the ***Front view button*** again. If you look at the picture above you will find that some of the lines become red when you select something. These lines are called *Control Rulers*. A surface can have up to 64 such control rulers, and each control ruler can have anything between 1 and 256 ***control points***. The ***green dot*** is always located at the ***first control ruler*** and at the ***first control point***. This is important to know when editing models. In this case the green dot is at the bottom left corner of the surface.

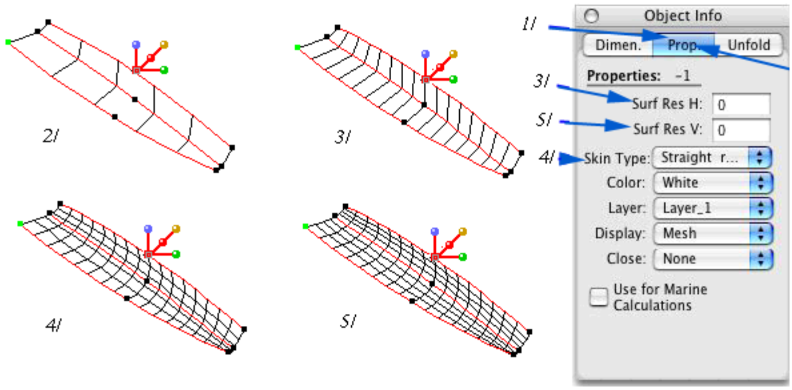
Now choose the first and second control points in the second and third control rulers by Shift-clicking on them (**1**). They should have red dots if correctly chosen. One of the points will carry the ***Intersection Point*** marker. Change view to the ***Side/Right*** view (**2**). Click on the ***Intersection point*** and drag it horizontally to the right (**3**). Deselect by clicking beside the surface and draw a selection rectangle around the control points in the third ruler. Drag the selection a little further to the right (**4**).

Adjusting the surface resolution

We now have something that looks like a boat hull even though it is fairly rough and "edgy". Let's experiment with the parametric features of TouchCAD. Deselect by clicking outside the model with the ***Selection Arrow*** tool. Choose ***Fit to Window*** in the ***Page Menu*** (or use Command/Control 4). Select the **Iso Front Right** view by pressing the "**3**" key on the numerical keyboard. Click on one of the control points (**2**). This enables us to edit the properties of the surface. Click on the "*Prop.*" Tab on the ***Object Info*** palette (**1**). Change the value in the field ***SurfRes H*** (horizontal) from 5 to 16 (**3**). The surface is now subdivided into 16 steps, so this field controls the number of steps along the ruler.

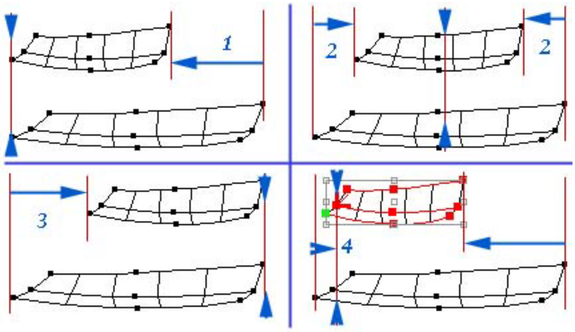
Changing surface type

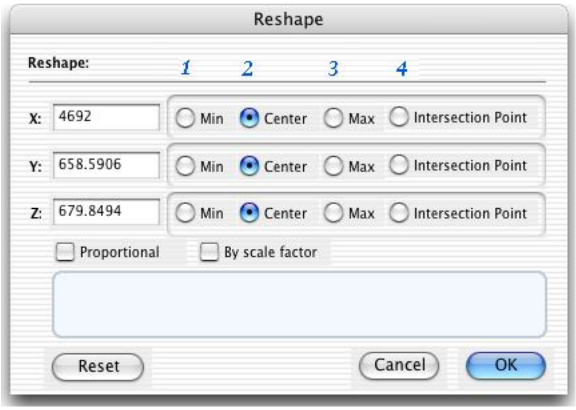
At present the surface is defined as having straight lines perpendicular to the rulers. Change this by changing the ***Skin Type*** popup ***to Smooth Along Ruler /Smooth Perpendicular to ruler*** (4). The surface is now displayed with 16 steps along the ruler and 5 perpendicular to the ruler. Increase the resolution perpendicular resolution in the ***SurfRes V*** (Vertical) field to say 8 (5). The surface now has a higher resolution in the perpendicular direction.



Resizing the model.

The boat hull is at present just a shape with no particular size defined. In order to scale it to a real physical size we simply select the surface and then select the ***Reshape*** command in the ***Edit*** menu.





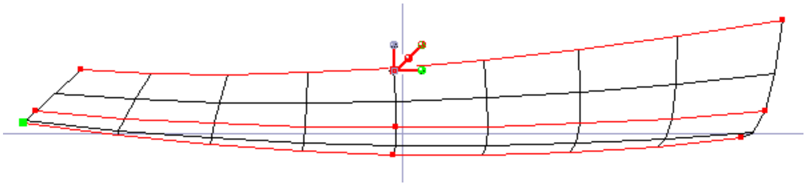
Specify the size of selection as follows: X = 6000 (mm), Y = 1200 and Z = 1200. Scale from the center in the X and Y directions and from the lower side (min) in the Z direction. Click OK. The surface is now 6000 x 1200 x 1200 mm.

Being able to choose a point from which to scale is useful in this case. The Reshape dialog enables you to scale from the ***minimum point (1)***, ***center point (2)***, ***maximum point (3)***, and ***Intersection point (4)*** in each respective direction. The picture above illustrates the ***X direction*** as seen from the ***Front*** view.

In the X and Y directions it was a good idea to scale from the center whereas in the Z direction it was better to scale from the ***Min*** point because it is located at the center line. The proportional option is used if you want to keep the proportions of the selection.

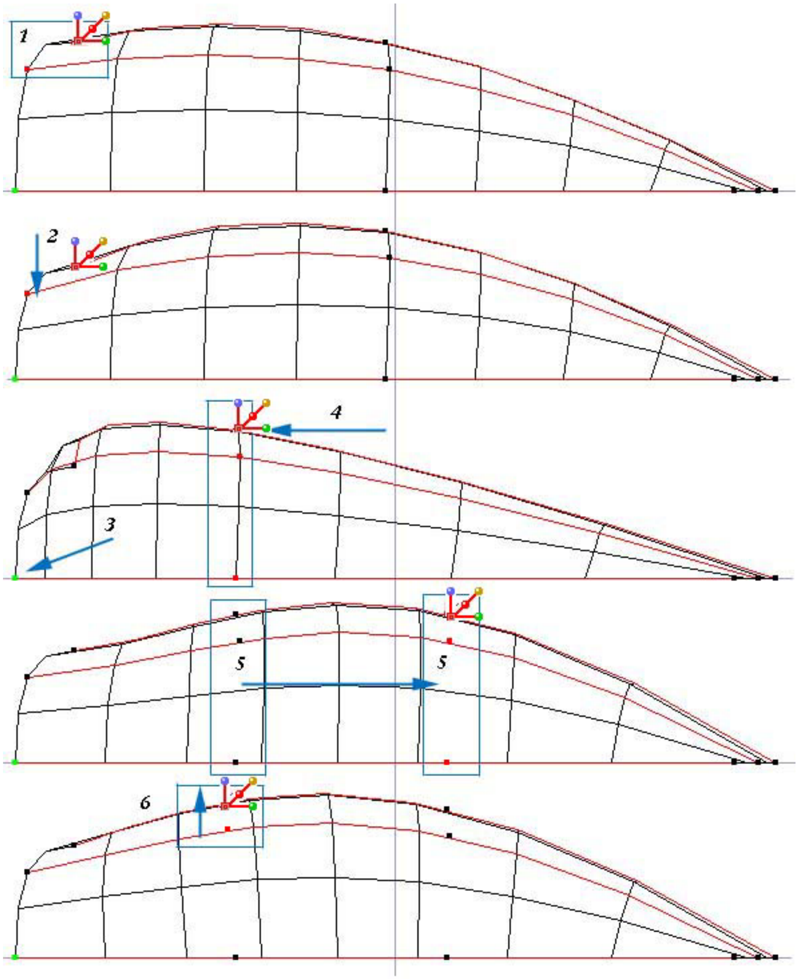
Adjusting the drawing scale

The model is now probably too big to fit into the drawing area if you use a letter sized printer. It is easily fixed by changing the drawing scale. Choose Drawing Scale… in the Page menu. Change the scale to 1:25 and click OK. Choose Fit to Window from the Page menu. Move the model so that it is similar to the picture below, which is probably how it would float.



Adding more control points.

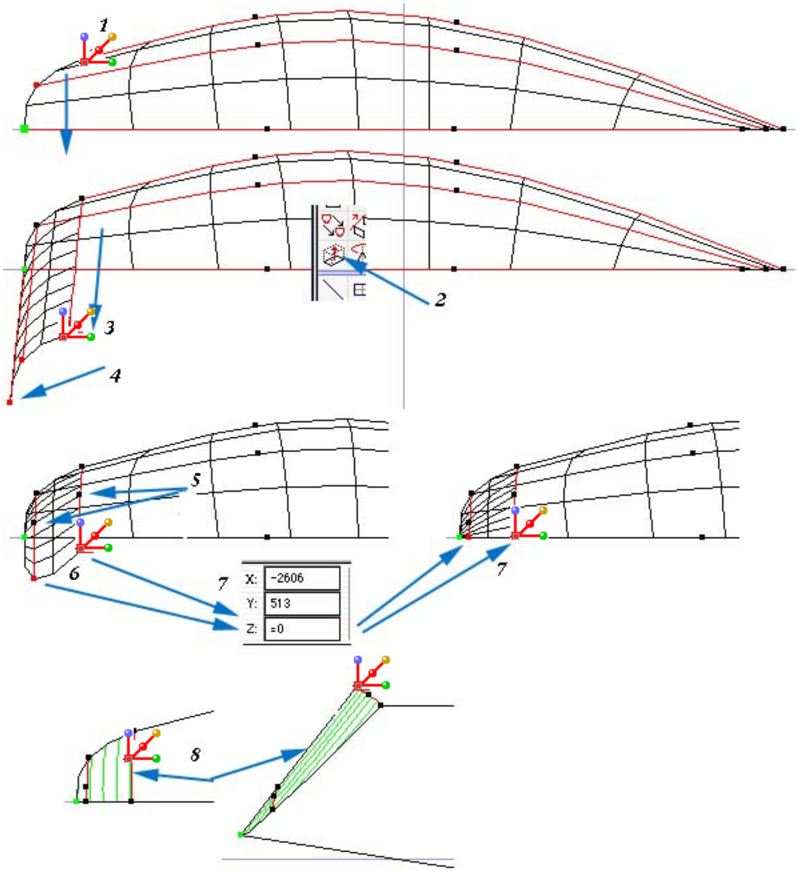
Change view to the *Top* view. We will now add some control points to the surface to get a better control of the shape.



Adjust the width of the transom as seen above (**1** & **2**). Locate the green point of the surface. TouchCAD always ***adds control points and rulers upwards and away from the green starting point.*** Move the three center points to the left to add some more space (**4**). Press the ***Alt/Opt*** key and then start dragging to the right. Three new points occur (**5**). Adjust the shape of the model (**6**). You probably need to adjust the shape from the Front and Side views as well. You can delete any number of points by selecting them and then press ***Backspace*** or ***Delete***. You can ***add a new control ruler*** (red lines) by selecting a point and then press ***Shift + Alt/Opt*** before starting to drag. The dragging direction is always ***away*** from the control ruler containing the green dot.

Adding a transom.

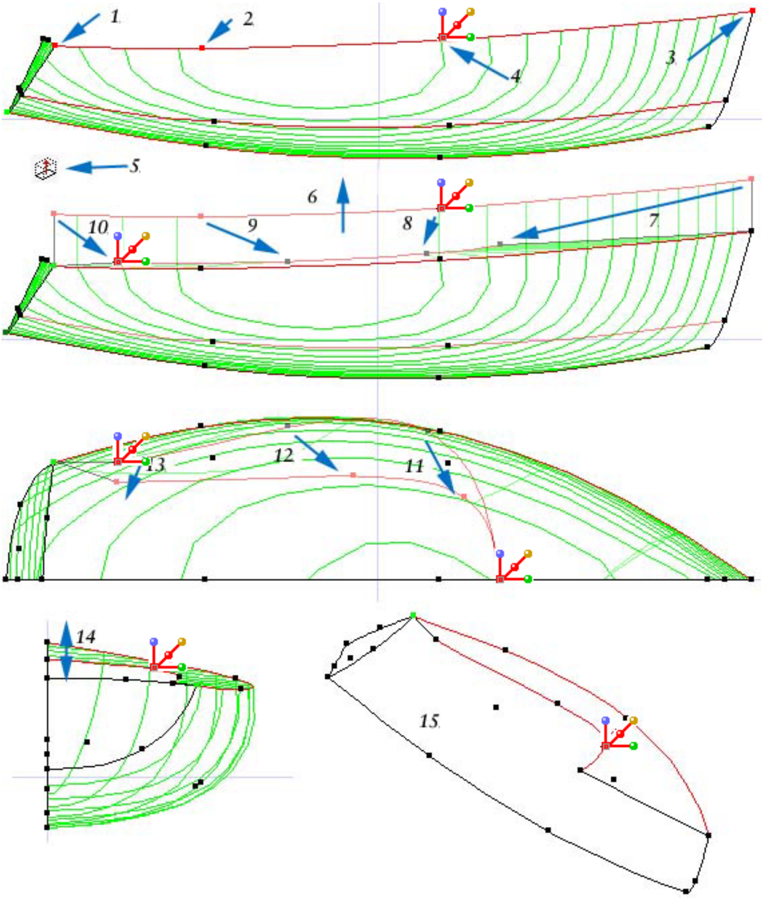
The next step is to add a transom. This is best-done using the ***Extrude*** tool. The picture below shows the steps.



Select the three control points to the left or stern of the boat (**1**). Select the ***Extrude*** tool in the ***Tool*** palette (**2**). Click on one of the selected control points to activate it and drag downward *(3)*. Select the ***Selection Arrow*** in the ***Tool*** **palette** and deselect. Select and delete the far-left point *(4)* (we only need one point in this ruler). Select the other two points and move them upward (**5**). ***Alt/Opt-Drag*** downward to add two new control points (**6**). Click on the ***Z field*** above the drawing window and type "**=0**" and then press ***Enter*** (7). The two selected control points coordinates both become zero by this action (7). Change display mode to ***Cross Sections*** in the ***Object Info*** palette and adjust the lines so that the green section lines roughly aligns with the edge lines seen from the side (**8**).

Adding a deck.

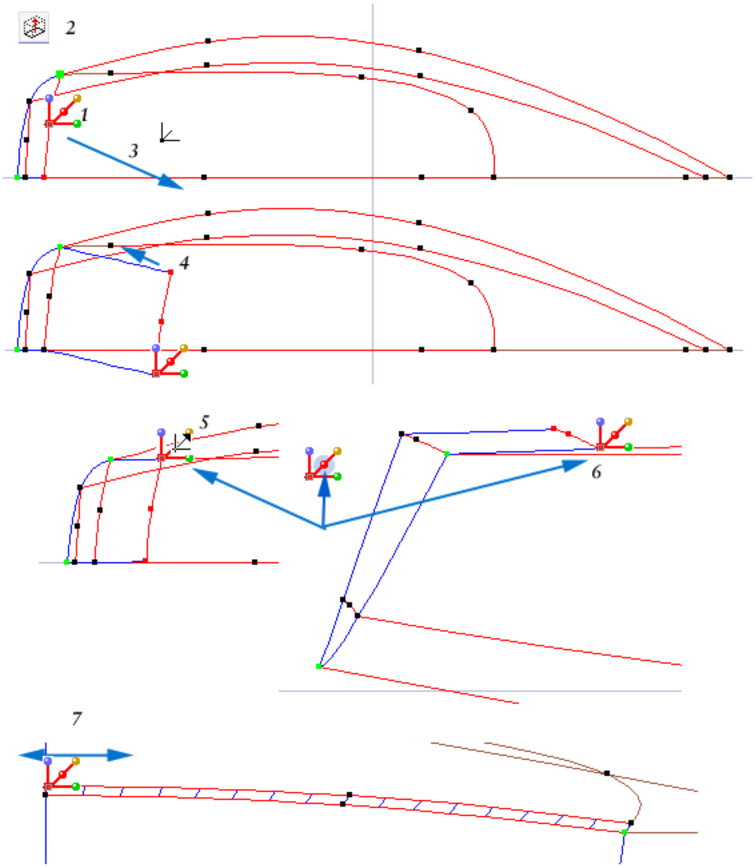
In the next step we will add a deck to the model. It is essentially done in the same way as was used when adding the transom. The pictures below show the steps.



Select the four control points along the sheer line (**1,2,3,4**). Technically, you only need to select two points along the ruler to specify that you want to extrude from the chosen edge of the hull surface. Another thing to note is that the Intersection Point (***4***) is placed in the middle of the surface before extruding. Placing it along the edge between the hull and transom may create an unpredictable result because it connects to two surfaces. It is therefore better to place the Intersection Point in the middle. Select the ***Extrude*** tool in the ***Tool*** palette (**5**). Click and drag upward (**6**). Move the inner edge points to their new location (**7, 8, 9, 10**). Select the ***Top*** view. Move the edge line control points upward and downward to get roughly aligning cross sections (**14**). The final deck is seen in the right isometric front view (**15**).

Joining surfaces.

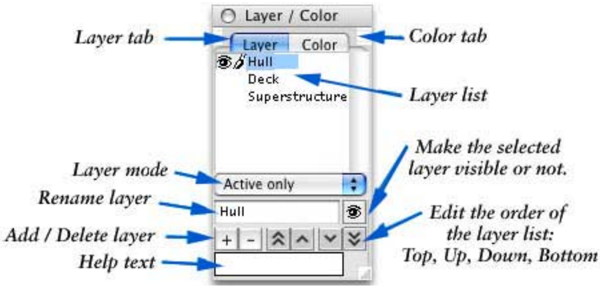
Based on the knowledge of snapping it is now easy to add an afterdeck and join it with the other deck element. Select the edge points along the upper side of the deck by Shift-clicking on them (**1**). Select the Extrude tool in the Tool palette (**2**). Click on one of the edge points and drag forward (**3**). Switch to the Selection Arrow tool and join the edges of the two deck elements (**4**). Use the Move 3D feature (item 7 on the previous page) if required to join the two surface edges (**5**). Changing view to a horizontal view verifies that the 3D snapping has occurred (not really required) (**6**). Move the point closest to the center point to Z=0 (**7**). Adjust the shape if required.



Working with layers

When you continue to add parts such as a cockpit and so on, it may be convenient to locate the surfaces onto separate layers. It makes the model less cluttered when editing. It also makes it easier to select and edit individual points.

Locate the ***Layer/Color*** palette. If you can't find it, try choosing ***Layer/Color*** palette in the ***Palette*** menu. Click on the ***Layer*** tab if it is not already selected. The picture below shows the elements.



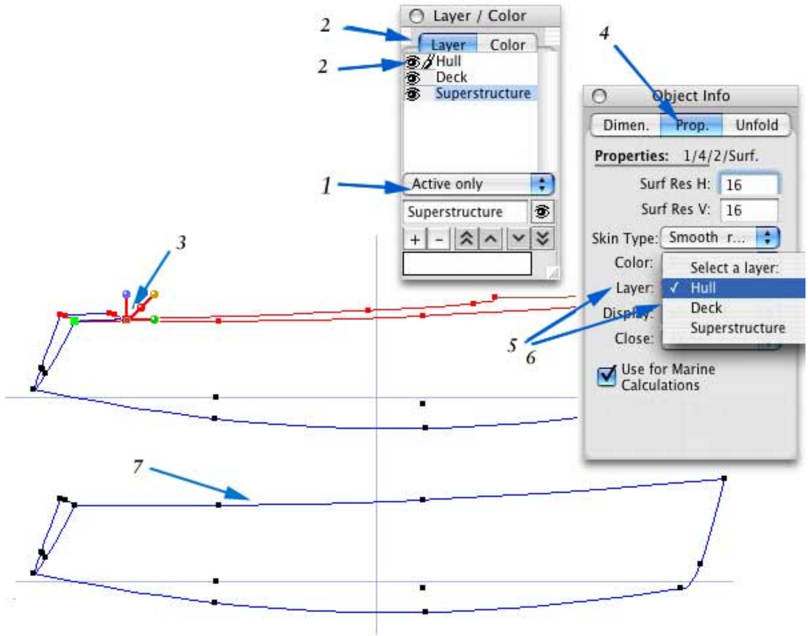
The two symbols in front of the layer name indicate that the layer is visible *(the eye)* and that it is the active layer *(the brush)*. You activate a layer by clicking on the ***layer name*** (if more than one layer is available). The brush symbol then moves to that layer. A layer is made invisible by clicking on the layer name and then clicking on the Eye button further down to the right in the palette. It is an on/off button. The objects in the layer are still visible because the layer is still the active layer. It overrides the invisible command. The invisible layer becomes invisible when you click on another layer.

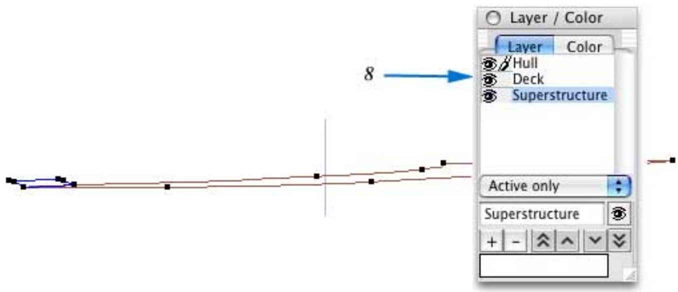
Another factor to consider is the ***Layer Visibility mode pop-up*** (also found in the ***Organize*** menu). The ***Active Only*** Option only displays the active layer regardless of whether the rest of the layers are visible or not. The ***Show Others*** option displays all layers defined as visible but does not allow snapping to objects in other layers. ***The Show Snap Option*** displays all visible layers but does not allow editing of other layers. ***The Show Snap Modify Others*** option allows you to edit any layer defined as visible.

Now, add two layers by clicking twice on the ***Add Layer*** button (+). Click on the layer named Layer-1 to activate it and then on the Rename field. Name it "***Hull***" and then click on the name in the list to verify. Rename Layer-2 to "***Deck***" and Layer-3 to "***Superstructure***".

Moving objects to other layers.

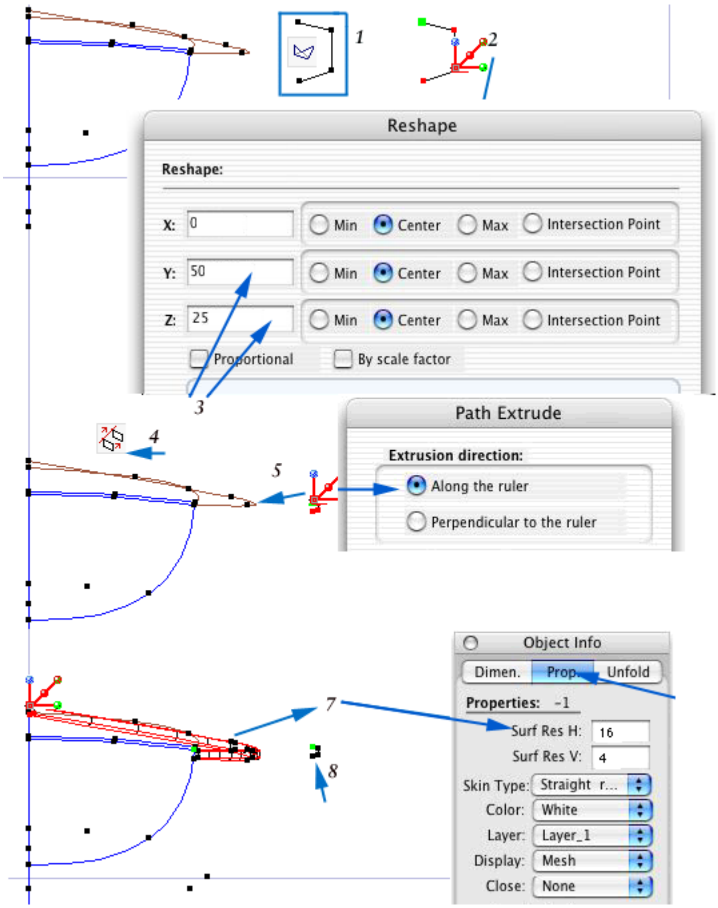
Check that the ***Layer Visibility*** in the ***Layer*** palette is set to ***Active Only*** (**1**). Click on the "***Hull***" layer to activate it (**2**). Click on a control point that exclusively belongs to one of the deck surfaces to select it (**3**). Locate the ***Object Info*** palette (**4**). Select ***Object Info*** from the ***Palettes*** menu if you can't find it. Click on the ***Properties tab (Prop.)*** (**4**). Locate the ***Layers popup*** (**5**) and select the "***Deck***" option (**6**). The selected surface disappears though it is not really the case (**7**). It just moves into another layer. This layer is currently invisible because the ***Layer View*** mode is set to ***Active Only*** (**1**). You can make the deck visible again by selecting the "***Deck***" as the active layer (**8**), or by selecting anything below ***Active Only*** in the ***Layer Options*** menu (**1**).





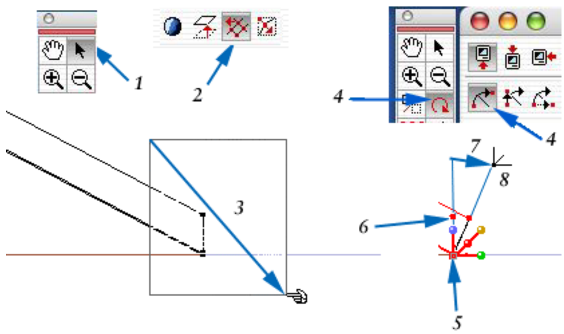
Path Extrude.

Now, let's add a gunwale molding along the sheer line of the hull. It is best done using the Path Extrude tool in the tool palette. Before adding we need to create a cross section of the intended molding. Select a view where you see the hull from the front view, in this click on the right view button or press the "6" key on the numerical keyboard. Start by drawing a basic shape using the ***Polygon*** tool in the ***Tool*** palette (1). Adjust the size using the ***Reshape*** command to say 25 x 50 mm (2-3). Select the Path Extrude tool in the ***Tool*** palette (4). Click on the appropriate point in the cross section and draw a help line to a control point along the sheer line ruler of the hull and then click again (5). Select ***Along the ruler*** in the ***Path Extrude*** dialog (6). The ***Perpendicular*** option extrudes the section down to the centerline of the hull in case you wonder. Adjust the resolution of the new surface to the same resolution as the hull (7). Delete the cross section original if necessary (8).

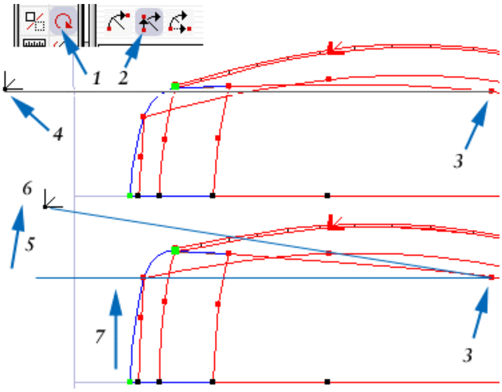


Using the Rotate tool.

When examining the path extrusion generated on the previous page it becomes evident that we need to adjust the new surface. Change view to the ***Top*** view. ***Zoom*** in on the bow of the hull where the path extrusion ends. Let's adjust the end points of the extrusion so that they are roughly perpendicular to the hull. Start by drawing a selection rectangle around all control points in the area (3) using the ***Selection Arrow*** tool (1) set to ***Vertex Mode*** in the ***Selection Mode*** bar (third form the left) (2). It enables us to select individual control points. Select the ***Rotate*** tool in the ***Tool*** palette (4). The action consists of three steps. Click on the center point of the rotation (5), which is the control point of the hull. Click a second time on the control point located at the greatest distance away from the first point (this is the most practical option in this case) (6). The selected control points start to move with an imaginary lever that now occurs. Move the cursor until the control points are roughly perpendicular to the hull (7). Click a third time to finish the rotation session (8).

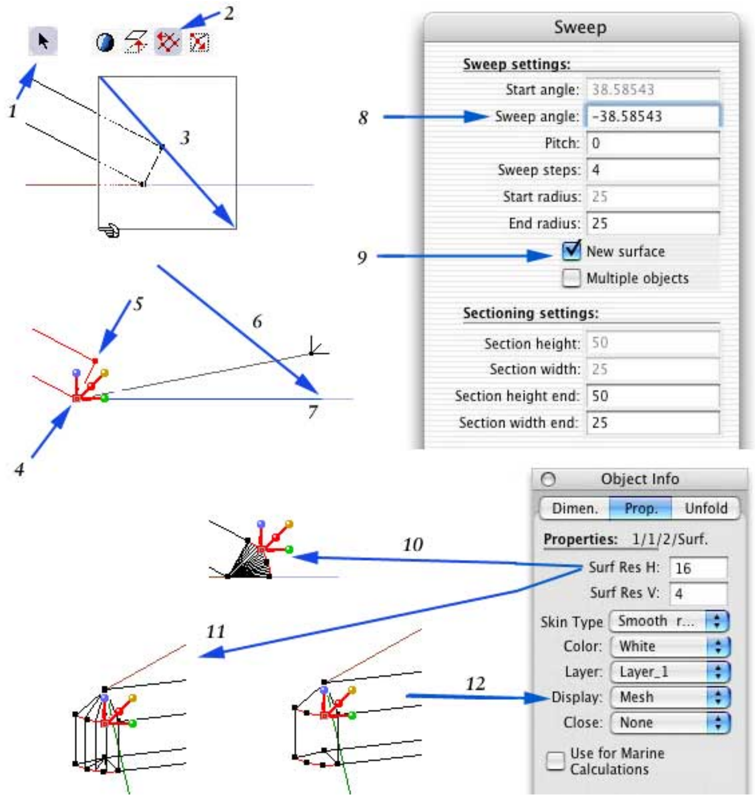


Though not applicable in this case, there are other useful options in the ***Mode bar*** (2). The options are from left to right, ***Normal two-directional rotation***, ***Constrained Horizontal*** and ***Constrained Vertical*** rotation. The later two constrain in one direction (X or Y) but performs the calculation in the other direction. The example below shows where we want to make the rear end of the hull gradually wider as we move towards the left. We also want to keep the horizontal coordinates intact. Start by selecting the appropriate control points. Exclude the points along the centerline because we don't want to move them. Select the ***Rotate*** tool in the ***Tool*** palette (**1**). Click on the ***Vertical Constrain*** option in the ***Mode*** bar (**2**). Click on the center-point of the rotation (**3**). Move the cursor to the left and click a second time to establish a second point used to define the lever (**4**). Drag upwards to rotate (**5**). Click to finish the rotation session (**6**). The after end is gradually wider (7), but keeps the width closer to the rotation point (**3**).



Using the Sweep tool.

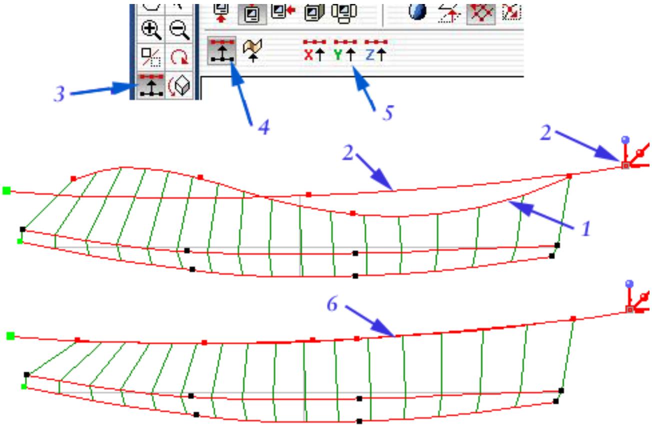
When moving back to the front end of the extrusion, it may be a good idea to add a radius around the bow of the hull. This can easily be done using the ***Sweep*** tool in the ***Tool*** palette. These are the steps:



Select the ***Top*** view and then zoom in so that you clearly see the end of the extrusion (in the Bow). Use the ***Selection Arrow*** tool (*1*), ***Mesh*** selection mode (*2*), to draw a selection rectangle around the end of the extrusion (*3*). The ***Sweep*** procedure is similar to the ***Rotation*** tool. Click on the center point of the sweep (*4*), on the other end of the extrusion end to create a lever (*5*), rotate the lever down to horizontal (*6*), press the Shift key to lock to a horizontal constraining line and then click to finish the sequence (*7*). The ***Sweep dialog*** occurs. Check that the Sweep angle is reasonable (*8*) and that you will create a new surface (*9*) (yes, you can add a sweep within a surface). The new surface automatically inherits the properties of the surface it starts from, and in this case the horizontal resolution is 16 steps. This may be practical on a long object, but too many in this case. Select a control point in the new surface, and change the resolution to say four steps (***10-11***). (***11***) Shows the surface displayed as a mesh and (***12***) as a contour.

Using the Align tool.

The Align tool aligns objects to curves or surfaces. Below is a picture showing the steps.



1/ The curve needs some adjustments. Dragging points can of course do this but in some cases it is easier to use a control ruler, especially if there are lots of points to be moved.

2/ Draw a temporary curve using the Polygon tool and make sure that it has a suitable resolution for the job. If not, adjust it in the Object Info Palette. Note that you need a curve that moves along a ruler to make it work. Select all points to be moved and then Shift-click on one of the points along the control curve. The Intersection point should be located on the control ruler because this tells TouchCAD which curve to use and which points to move.

3/ Select the Align tool in the Tool palette.

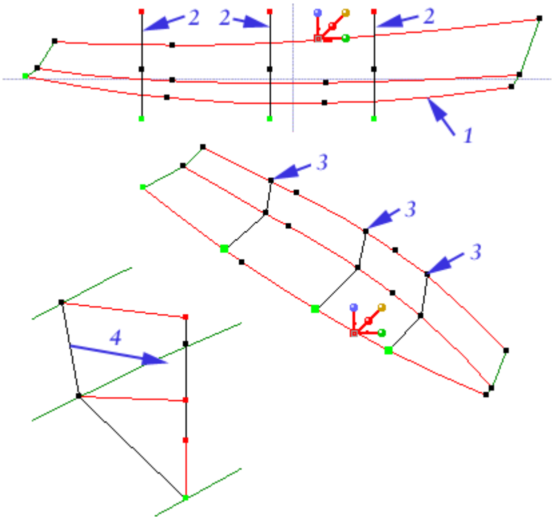
4/ Make sure that the Align to ruler button is selected in the mode bar.

5/ Locate the relevant directional button in the mode bar and click on it. In this case we wanted to move points in the Y direction and we therefore used the Y button.

6/ All selected points move to align with the selected curve, in this case in the Y direction.

Generating a bulkhead

Below is another example where rulers in a surface are used to create three bulkheads.



1/ The surface used for creating the new elements.

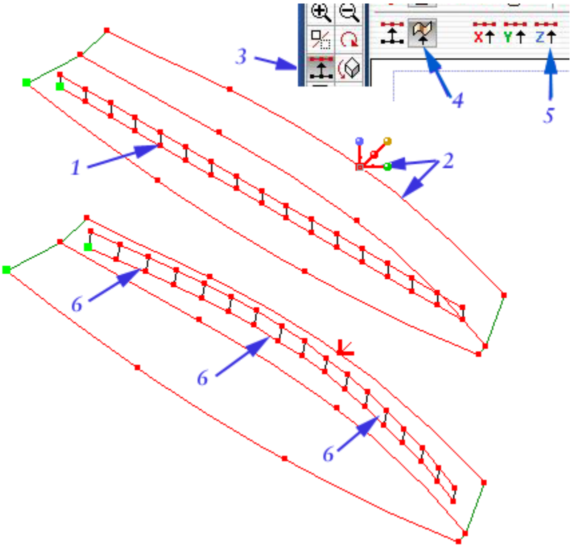
2/ Draw a polygon, in this case having three points, because we have a three-ruler surface with straight lines in between. Select the three upper points and then place the Intersection point on the upper ruler by Shift-clicking on it. Use the Align tool as described on the previous page. Note that there are two directions to align to here and it is usually easier if you change view in between the two steps to see what you are doing. Repeat the steps for the center and lower rulers.

3/ This is what the finished result should look like.

4/ Select all points in the three new curves. Change view to see them as cross sections and make sure that the Vertex mode is selected in the Selection mode bar. Press the Option (or Option-Shift depending of the direction of the polygon) and drag. This adds new points to the curve and converts it from a curve to a surface. Adjust the points to generate the frame or bulkhead.

Adding a stripe

Below is an example of use of the Align tool and illustrates the use of the Align to Surface mode. The mode moves points to align to a surface instead of a ruler.



The object of this exercise is to create a stripe along the surface of the hull.

1/ Start by drawing a rectangle surface using the Rectangle tool in the Tool palette and using the Surface mode in the Mode bar. The rectangle has four control points but we can easily add more points by increasing the resolution to say 16 steps along the rulers, using the Object Info palette, and then add points using the Controls at Surface Resolution command in the Edit menu. Make sure that all points are selected in the new surface.

2/ Shift-click on one of the control points of the hull surface. This tells TouchCAD to use the hull as the master surface.

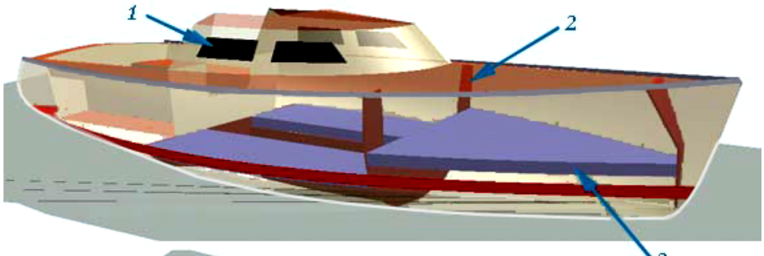
3/ Select the Align tool in the Tool palette.

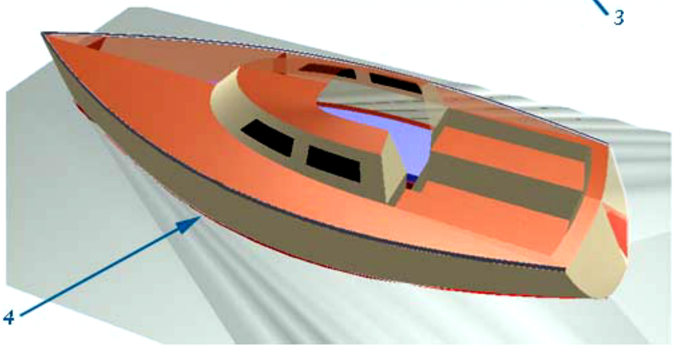
4/ Select the Align to Surface button in the Mode bar.

5/ Click on the relevant directional button, in this case the Z button. All selected points project to the surface of the master surface, in this case the hull.

A few tips for using the Align to Surface tool

In some cases it appears that some points refuse to move to the surface. If so, check that the point is located inside the projected area, by simply looking at it from the relevant view. If you intend to render the aligning and master surfaces together it is usually best to move the aligning surface away from the carrying surface a small distance to avoid zebra-style renderings. You will understand why we call it that when you see it.





More Align to Surface examples

Above are more examples showing the use of the Align to Surface tool.

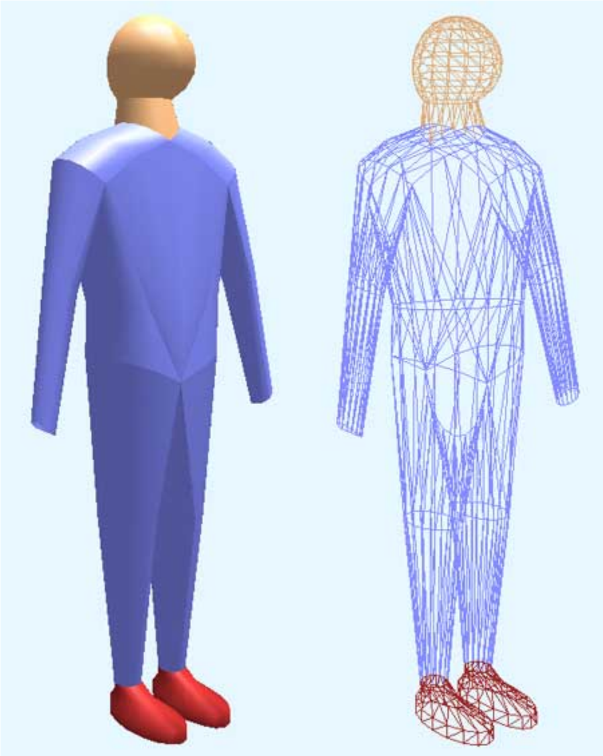
The ***glass surfaces*** *(1)* where projected on the cabin side. Note that the surface was nudged away a bit from the cabin side to get it to render well. If it is too close you typically get a zebra style flicker when rotating it and that means that the two surfaces are too close to one another. A copy was placed on the inside in a similar way to indicate the location of the glass seen from the inside.

The ***bulkheads*** *(2)* where drawn as a double line surface where the distance between the lines represent the true thickness of the bulkhead. Change the surface resolution to 8-16 steps along the rulers and then run the Controls at surface resolution command to add more control points. Project the points to the hull surface using the Align to Surface tool. Extrude the surface towards the centerline using the Extrude tool in the Tool palette (Solid Extrude mode). Select the Ignore Group mode in the Edit Mode and adjust location of the inner points to get the desired shape. An interesting side effect with using a solid bulkhead is that it enables you to calculate the edge angles along the hull edge side, which of course varies along this edge.

The ***cushion*** *(3)* example is similar to the bulkhead, except for the horizontal orientation. The thickness of the material and the location in the hull makes it even more important to use a double-sided projection along the skin of the hull because the angles can sometimes be very steep, especially on sailboats.

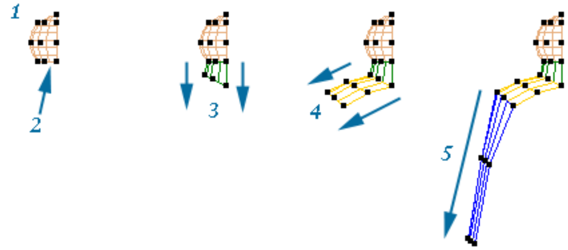
The ***water*** *(4)* example shows a slightly wave shapes ruler drawn along the waterline of the hull. Extrude the surface sideways / backwards and away from the hull and then project the inner edge to the hull surface. This generates a wave style water surface.

Tutorial, a human shape



Modeling a human shape

Modeling a human shape or other similar shapes may sound difficult, but it is mostly a question of having a good strategy. Once you have that, it is mostly a matter of time and experimenting back and forth to build a good looking model. This example briefly illustrates the basic steps of one way of doing it. It also illustrates how to build models consisting of connecting surfaces, which is a common modeling technique used when modeling with TouchCAD. Note that this strategy can also be used for modeling animals, because they also have a similar configuration with elements that stick out from the basic torso in one way or another.



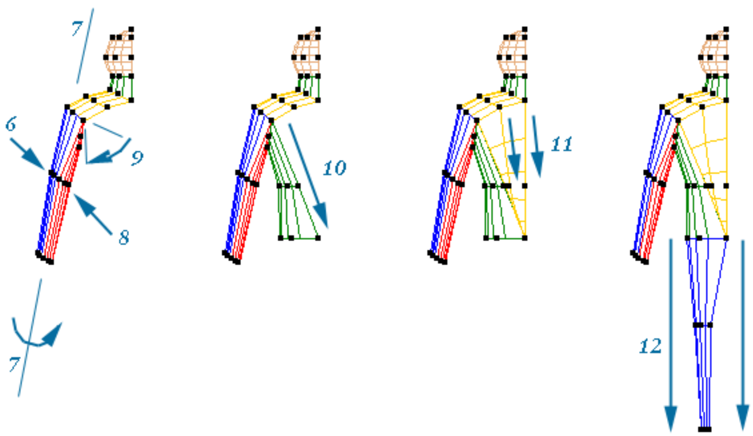
1/ Start by double clicking on the sphere tool in the Tool palette. Specify the dimensions and that you want a 180 degree sweep. The reason for this is that it is easier to model only ones side of a symmetrical object such as a human body.

2/ Delete the lowest control points so that a semi circular opening occurs at the bottom.

3/ Select the lowest control points and use the Extrude tool in the Tool palette to extrude a neck. Use one step in the Extrusion dialog. Rotate the lower control points a bit as seen on the picture.

4/ Use the Extrude tool again to extrude the shoulder piece. In this case it is better to use two steps in the Extrude dialog to allow a smother transformation towards the arm.

5/ Continue extruding the outer side of the arm downwards using two steps in the Extrude dialog. Don't worry too much about the shape yet. It is usually easier to adjust the proportions later on when the entire model is visible.



6/ Select the outer arm.

7/ Use the Mirror and Duplicate tool in the tool palette to generate the inner part of the arm.

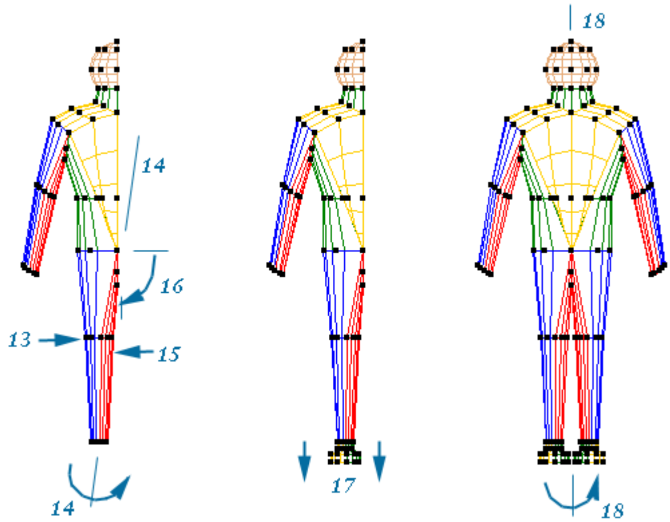
8/ Adjust the edges so that they snap to the corresponding control points on the outer side.

9/ Rotate the upper control points of the inner piece of the arm so that they are more or less vertical by using the Rotate tool in the tool palette and rotating from the joint between the outer and inner pieces.

10/ Select the rotated edge and use the Extrude tool in the tool palette to generate the side of the torso. Use two extrusion steps in the Extrude dialog. Additional steps can be useful if you want to build a more realistic shape, but as always, it is recommended to use a few steps as possible.

11/ Use the Extrude tool to generate the front and back sides of the torso, starting from the shoulder. Use the same number of extrusion steps as in the side of the torso. Make sure that the front, back and side edge points snap together and that the edge along the centerline is completely vertical to allow a perfect alignment with the other side of the shape that will be created later on.

12/ Select the lower edge control points of the side of the torso. Use the Extrude tool in the tool palette again to generate the outer side of the leg. Use at least two steps to allow leg bending around the knee joint.



13/ Select the leg.

14/ Use Mirror and Duplicate tool in the Tool palette to generate the inner side of the leg.

15/ Move the respective edge points so that they snap on the respective edges on the outer side.

16/ Use the Rotate tool in the Tool palette to rotate the upper edge control points of the inner side of the leg into a vertical orientation. Make sure that all control points along the centerline have the same coordinate setting. A quick way of doing this is to select all control points along the centerline, select the relevant coordinate field above the drawing window and there insert an "=" before the coordinate. Press Enter and all selected control points get the same coordinate value setting.

17/ Select all control points along the lower end of the leg. Select the Extrude tool in the tool palette and extrude the foot. In this case you probably need three extrusion steps to form a reasonably realistic looking shape. Note that you now extrude two shapes at the time and in order to do that you need to activate the Multiple Surfaces checkbox in the Extrude dialog. If not, TouchCAD only allows one extrusion at the time.

The foot shape now looks like a tube. This is however easily fixed by selecting the lower two sections, changing view so that you see the model from the side, and then adjusting the shape with the gray Resize box controls.

18/ Select all parts and use the Mirror and Duplicate tool in the Tool palette to create the other side.

Conclusion

The model is by no means ready now and probably needs lots of adjustments and tweaking to get the right proportions, dimensions and poses. The idea with this exercise is primarily to suggest a possible strategy.

Importing files

TouchCAD can read files exported from a other programs that supports the same file types as TouchCAD. There are however some things to consider:

It is usually best to simplify the imported data as much as possible. You can for example import thousands of triangles and then use the skinning features to build real TouchCAD surfaces but that is normally a fairly inefficient way to do it. TouchCAD allows you to control the number of mesh points used, so an excessive number of imported points are just a waste of space.

Importing a few polygon curves or just points and use the lofting features to rebuild the surfaces is usually better than importing lots of triangles because triangles needs more memory to define the same shape. In most cases you do not an improved accuracy by importing an excessive number of points.

DXF

Most CAD and 3D modeling programs support DXF export and TouchCAD supports the following import data types: Lines, circles, arcs, polylines, LW polylines, 3D faces, points, layers, 256 colors and anonymous blocks (but not blocks).

The DXF format is usually text based and this allows you to create your own DXF files using any word processor or by writing your own program that generates such a file. A typical minimal DXF file consists of the following elements (a four sided 3D polygon):

The file:	**Comments:**
0	This is the Header section. It starts here
SECTION	Copy this
2	Copy this
HEADER	Copy this
0	Copy this
ENDSEC	Copy this
0	Copy this
SECTION	Copy this
2	Copy this
BLOCKS	Copy this
0	Copy this
ENDSEC	Copy this
0	Copy this
SECTION	Copy this
2	Copy this
ENTITIES	..and ends here. Just copy this section.
0	This command starts the first polygon
3DFACE	This command indicates a closed polygon
8	Copy this

UNTITLED-1	Copy this (Layer info)
10	Copy this (Code for first X value)
800.000	First X value
20	Copy this (Code for first Y value)
1150.000	First Y value
30	Copy this (Code for first Z value)
250.000	First Z value
11	Copy this (Code for second X value)
650.000	Second X value
21	Copy this (Code for second Y value)
2100.000	Second Y value
31	Copy this (Code for second Z value)
400.000	Second Z value
12	Copy this (Code for third X value)
2950.000	Third X value
22	Copy this (Code for third Y value)
2650.000	Third Y value
32	Copy this (Code for third Z value)
300.000	Third Z value
13	Copy this (Code for fourth X value)
3450.000	Fourth X value
23	Copy this (Code for fourth Y value)
1100.000	Fourth Y value
33	Copy this (Code for fourth Z value)
100.000	Fourth Z value
0	End of polygon
ENDSEC	Copy this (closing commands)
0	Copy this (closing commands)
EOF	Copy this (closing commands)

You can also learn more about DXF files by opening a DXF file exported from TouchCAD or from another CAD program. A more detailed description of the DXF format can be found at AutoDesk's web site (the manufacturer of AutoCAD).

Import as -> VectorWorks

The ***VectorWorks import*** command import files from ***Nemetschek's VectorWorks and MiniCAD*** (the older name for VectorWorks) in the ***VectorScript*** file format (called ***MiniCad Text*** in MiniCad 7 and older versions).

TouchCAD supports the importing of the following VectorWorks ***data types***: Lines, circles, ovals, arcs, rectangles, 2D polygons, 3D polygons, 2D & 3D locus points, layers and fill / line colors.

The VectorScript format is fairly easy to use and read and is therefore well suited for building your own files to be imported into TouchCAD.

This is a small example showing a minimal syntax for importing a VectorScript file (two 3D polygons):

Poly3D(	Copy this (Start of 3D polygon)
800.000, 250.000, 1150.000,	X,Y,Z values - first vertex
650.000, 400.000, 2100.000,	X,Y,Z values - second vertex
2950.000, 300.000, 2650.000,	X,Y,Z values - third vertex
3450.000, 100.000, 1100.000	X,Y,Z values - fourth vertex
);	Copy this (End of 3D polygon)
Poly3D(	Copy this (Start of 3D polygon)
250.000, 200.000, 500.000,	X,Y,Z values - first vertex
800.000, 250.000, 1150.000,	X,Y,Z values - second vertex
3450.000, 100.000, 1100.000,	X,Y,Z values - third vertex
3750.000, -100.000, 300.000	X,Y,Z values - fourth vertex
); Copy this	(End of 3D polygon)

You can also learn more about VectorScript files by opening a VectorScript file exported from TouchCAD or from another CAD program. A more detailed description of the DXF format can be found at AutoDesk's web site (the manufacturer of AutoCAD).

Points

The ***Points*** import format is a ***simple text-based TAB based X, Y, Z, coordinate list***. The syntax is a s follows:

123.123 ***(TAB)*** 456.456 ***(TAB)*** 789.789 ***(Carriage Return)***

789.789 ***(TAB)*** 456.456 ***(TAB)*** 123.123 ***(Carriage Return)***

Etc.

Such a file can be generated using an ordinary word processor, spreadsheet or database program and the typical use is to import and make use of measurement points to create a model. The manual ***section*** concerning the ***Combine Into Curve*** command contains an example on how to process and refine points into usable surfaces.

Digitizing methods

A common question is whether it is possible to use various digitizing devices for converting physical models into a 3D model in TouchCAD.

Many 3D digitizers have the disadvantage of generating an excessive number of objects, and this is not very practical for use in TouchCAD unless the models have been reduced to a reasonable size by deleting excessive data.

MicroScribe range 3D digitizing arms

The MicroScribe range 3D digitizing arms and other similar devices can however be quite useful since the operator controls the input data. A human operator can easily detect where relevant changes in the model shape occur and the data provided can therefore be kept at a reasonable level. More info about the MicroScribe range can be found at www.immersion.com

PhotoModeler and other photo based digitizing programs

Another possible digitizing method is to use a photo based program such as the Eos Systems PhotoModeler program. Here a series of pictures from various views is compiled into a 3D model that con be imported into TouchCAD using the DXF, Wavefront, STL or VRLM formats. PhotoModeler is a Windows based program but rund well on Macs having Virtual PC installed. More info aboyt PhotoModeler can be found at www.photomodeler.com

Index

Lundström Design

Ekhagsvägen 7, 104 05 Stockholm, Sweden
Phone int+ 46 - (0)8 - 15 46 63
info@touchcad.com
www.touchcad.com