Touch-3D

Version 2.1

User's Manual

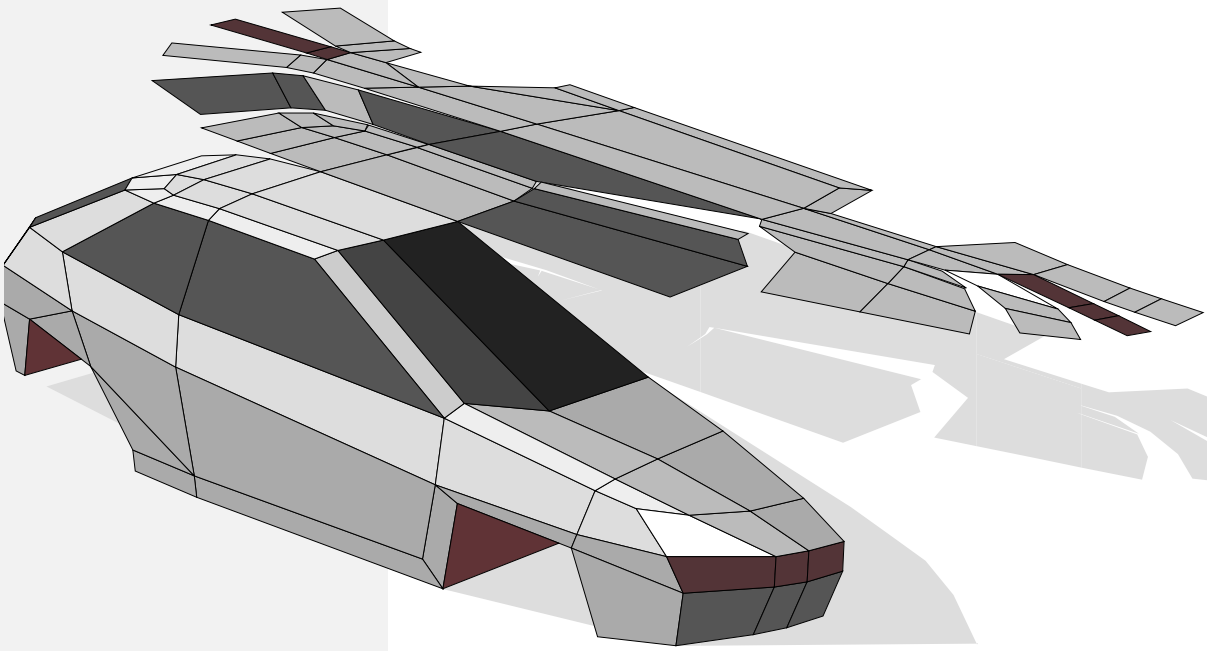


Table of contents

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 authorised 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, CURRENTNESS, OR OTHERWISE. THE ENTIRE RISK AS TO RESULTS AND PERFORMANCE OF THE SOFTWARE AND WRITTEN MATERIALS IS ASSUMED BY YOU. 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.

U.S. Government Restricted Provisions

If this Software is acquired by or on behalf of a unit or agency of the United States Government this provision applies. This Software: (a) Was developed at private expense, and no part of it was developed with government funds, (b) Is a trade secret of Lundström Design for all purposes of the Freedom of Information Act, (c) Is the "commercial computer Software" subject to limited utilization as provided in the contract between the vendor and the governmental entity, and (d) In all respects is proprietary data belonging to Lundström Design.

For units of the Department of Defence (DOD), this Software is sold only with "Restricted Rights" as that term is defined in the DOD Supplement to the Federal Acquisition Regulations, 52.227-7013(c) (1)(ii) and: Use, duplication, or disclosure is subject to restrictions as set forth in subdivision (c) (1) (ii) of the Rights in Technical Data and Computer Software clause at 52.227-7013.

Manufacturer: Lundström Design, Ekhagsvägen 7, 104 05 Stockholm, Sweden.

Phone int + 46 -(0)8 - 15 46 63, 15 47 77. Fax int + 46 - (0)8 - 15 82 85. E-mail: ludesign.@algonet.se

If this Software was acquired under a GSA Schedule, the U.S. Government has agreed to refrain from changing or removing any insignia or lettering from the Software or the accompanying written materials that are provided or from producing copies of manuals and disks (except one copy for backup purposes) and: (e) Title to and ownership of this Software and documentation and any reproductions thereof shall remain with Lundström Design, (f) Use of this Software and documentation shall be limited to the facility for which it is acquired, and: (g) If use of the Software is discontinued to the installation specified in the purchase/delivery order and the U.S. Government desires to use it at another location, it may do so by giving prior written notice to Lundström Design, specifying the type of computer and new location site. U.S. Governmental personnel using this Software, other than under a DOD contract or GSA Schedule, are hereby on notice that use of this Software is subject to restrictions which are the same as or similar to those specified above.

Notes and Acknowledgements

Touch-3D™ was written by Max Vizins and Claes Lundström. Documentation by Claes Lundström.

Registration & Updates

The Touch-3D™ master disks are warranted subject to the conditions of the License Agreement for a period of six months from the date of purchase by the end user. A completed Registration Card must be returned to Lundström Design to officially register your copy of Touch-3D™.

Only registered users are entitled to technical support, the Touch-3D™ newsletter, maintenance releases, and reduced cost upgrades.

Defective master disks are replaced free of charge to the end user for 6 months after purchase. Thereafter master disks will be replaced for a nominal service fee set by Lundström Design.

Lundström Design will make available, from time to time, upgrades to the purchased program for a nominal charge. Such upgrades, along with the original master copy of the program, shall be considered one program, subject in its entirety to the License Agreement.

All users of Touch-3D™ are invited to write us with suggestions and comments regarding possible features for, and uses of, Touch-3D™ as well as with any idiosyncrasies they encounter in their use of the program or any inconsistencies between the programme's performance and the documentation.

What's contained in the package.

In the Touch-3D™ package you should find the following items:

- Touch-3D™ program disk (1 & 2).
- Touch-3D™ introduction disk.
- Touch-3D™ user's manual.

Should you find any of these items damaged or missing, please contact your sales representative or Lundström Design.

Serial numbers

In order to get information on updates, our newsletter and technical support you should immediately return the registration card. Make sure to write down the serial number found on the master disk.

Technical support

In order to get access to our technical support you have to be a registered owner of Touch-3D™. If so, you can reach us by either calling us, sending a Fax, by letter, or e-mail. If possible, use fax or e-mail when you communicate with us. Our support department is normally open from 9.00-16.30 Central European time. The Fax machine is always on.

Installation

Insert the master disk. Drag the document called Touch-3D™ and the sample files to a suitable location on the hard disk. Double-click on the symbol to unstuff the files. Double-click on the programme to start it.

Hardware configuration

Touch-3D™ will work on any Mac equipped with at least a 68020 processor or better. It's highly recommended to use at a Quadra level Mac or better. We recommend at least 4 meg (megabytes) of RAM memory available for Touch-3D™. Touch-3D™ is supplied in two versions. The difference is that one of the versions requires a floating point processor and the other does not. Use the FPU version when possible because it provides better performance on 68K Macs. If you use a Power PC based Mac it does not matter which version you use.

System configuration

Touch-3D™ needs System 7.0 or later to work.

Major Concepts

If you are unfamiliar with the use of menus, windows, scroll bars or the concept of pointing, clicking, selecting, etc., it is recommended that you familiarize yourself with the Macintosh operating system before you continue with Touch-3D™. Your Macintosh came with a manual that explains these Mac operating standards and conventions. If you are unsure of these terms, refer to the Macintosh manual for more information.

Click

To position the pointer on an object and briefly press and release the mouse button.

Drag.

To position the pointer, press and hold down the mouse button, move the mouse and release the mouse button. You always drag objects on the screen.

Select.

To click on an object, icon, graphic object once, which will highlight (darken) it. A selected object will take the action of the next command.

Vertexes and polygons are selected by using the Selection Arrow tool found on the Tool palette. There are four selection modes, causing the Selection tool to behave in four different ways; Layer, Polygon, Mesh and Vertex mode. Layer mode is used for selecting the entire content of a layer. Polygon mode is used for selecting polygons as units. Mesh mode is used for selecting vertexes, and any connected vertexes as units. Vertex mode is used for selecting vertexes individually. You change Selection mode by clicking on one of the Selection mode buttons in the Mode bar.

Keyboard short-cuts.

Unlike many Macintosh applications, Touch-3D™ has keyboard short-cuts that may not have a modifier. For example, you can select the X coordinate edit field by pressing the ”x” key while you work in the drawing window. Notice that there is no modifier; you just type the single character. Other keyboard short-cuts are more conventional in that they use the Command key to modify the short-cut character. For example, to open a document, press the command key, and while holding it down, press the ”o” key.

Arrow keys.

The arrow keys are used for moving selected polygons or vertexes. Selected objects always move in the direction of the arrow, regardless of which view you are in. The "+" (plus) and "-" (minus) keys are always used for moving in the direction towards / away from you.

Polygons.

The polygon is the basic drawing element used in Touch-3D™. Each polygon must have either three or four sides, and the vertexes can be placed individually anywhere in the drawing space, in full 3D.

New polygons can be created by using the Polygon and Free Form polygon tools on the Tool palette, by the Create Object Mesh command found in the Edit Menu, by importing a Shape from the Shape folder, or by importing from other programs.

New polygons based on old polygons can be added by using the Copy, Paste, Duplicate and Duplicate Array commands found in the Edit menu, and by using the Cut tool, Radial Extrude (one of three sub tools in the Polygon tool), or Mirror Copy Tools on the Tool palette.

Two polygons can be merged into one, using the Join Tool on the Tool palette.

Polygons, and individual vertexes can be moved and modified by simply dragging objects on the screen, by typing in new coordinates in the Data Display bar, by using the Scale, Reshape, Move, or Rotation tools found in the Edit menu, and by the Rotate, Mirror, Alignment ruler tools found on the Tool palette.

Views.

There are five standard views; Front, Top, Side, Unfold, and Perspective view. You can change view by clicking on one of the view buttons found on the Mode bar.

The Front, Top, and Side views allow editing by any tool.

The Perspective view might be used for editing, even though we recommend the Front, Top or Side views for editing and drawing. The Rotate, Mirror, Mirror Copy, and Help Line tools do not work in this view. The view point can be moved by using the Rotate View tool found on the Tool palette.

The Unfold view allows selection of objects and vertexes, moving of parts by dragging, using arrow keys, or by entering new coordinates, rotation of parts using the Rotate Tool, and the use of the Cut, Join, and Radial Extrusion (Polygon sub tool) Tools on the Tool palette. When moving objects in the Unfold view, all vertexes belonging to the group will be moved.

All but the unfold view allows you to add a real perspective to the model. This is done by selecting the Perspective command in the Organize menu.

Working planes.

Working planes can be thought of as a transparent surface on which polygons are located when drawn. The Intersection point is used for placing the working plane in space. The coordinates of the working plane can be seen on the Data Display bar. In

the Front view, the working plane is located at the Z coordinate. In the Top view, the working plane is located at the Y coordinate. In the Side view, the working plane is located at the X coordinate. To determine the location of the working plane, look at the coordinates of the Data Display bar, while no vertexes have been selected. When the cursor is moved on the screen, only two coordinates change. The coordinate that does not change, represents the location of the working plane.

There are two ways to set the location of the working plane. In the first method you simply click on a vertex. This sets the Intersection point, and consequently the working plane, to a new location in the direction away/towards you. In the second method you click on the coordinate and type in a new value. If you want to move the working plane using this method, be sure to deselect all vertexes. Otherwise, you will move all selected vertexes, which may not be what you intended to do.

Another very useful feature with the working plane/Intersection point is that it provides a center point around which views are being rotated. To avoid "loosing" the model in the 3D space when you change view, be sure to click on a vertex to define the location of the Intersection point/working plane. If it's placed somewhere in the center of the area in which you want to work, and change view, you can immediately start working without any tedious scrolling and zooming to locate and center the model.

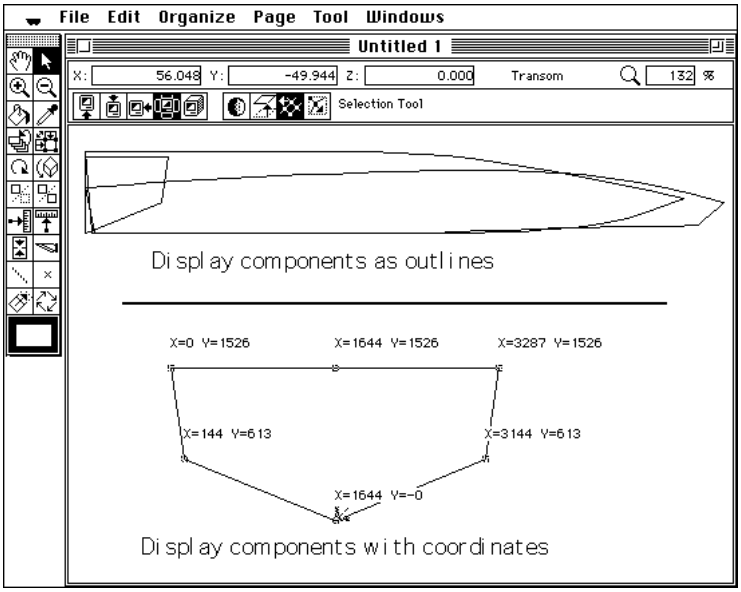
Unfolding.

The unfolding function in Touch-3D™ works like a view. Whenever you click on the Unfold view button, Touch-3D™ automatically calculates the unfolded shape of any 3D model found in the drawing. This means that you, interactively, can use a trial and error method for finding an optimal unfolded shape by making small changes in the 3D model.

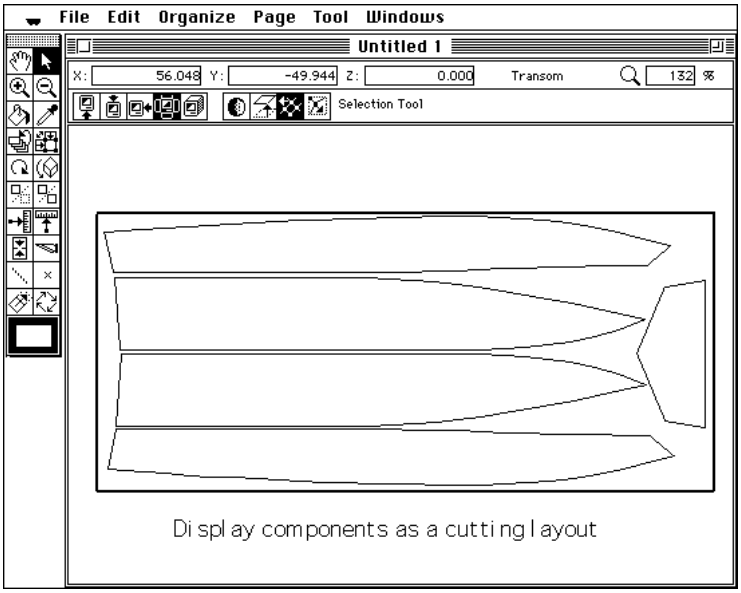
Another unique unfolding feature is that Touch-3D™ allows you to change the order in which models are being unfolded. This is done by using the Resort and Reconnect tools found on the Tool palette.

Unfolding Strategies and techniques.

The Unfold view allows you to move and rotate parts of the model as components to create cutting layouts to optimise the use of material. Two basic strategies can be used. In the first strategy each object is rotated and moved in such a way that each component aligns with the left/bottom sides of the coordinate system This is useful for generating a coordinate list drawing. The second strategy is used for generating a cut out layout from a single sheet of material where the object of the layout is to optimize the use of material.

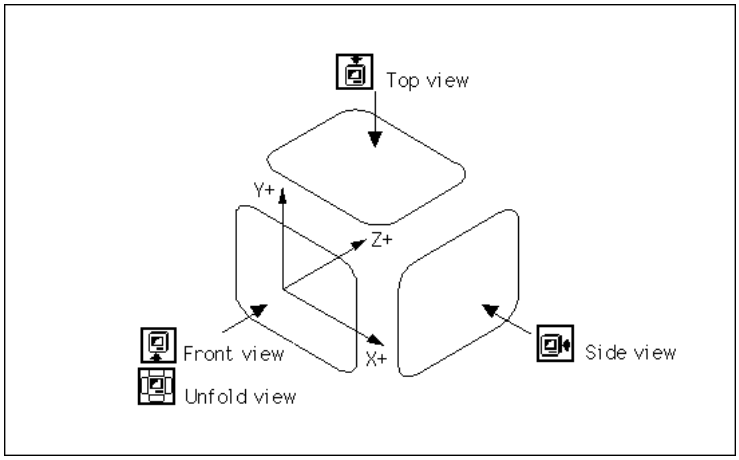


Above is an example showing the Outline and Show Coordinates option. In the upper example, all components have been aligned to the left and bottom at X/Y = 0. Note that you only get coordinates for selected vertexes. Below is an example where the components have been fitted into a single sheet of material.



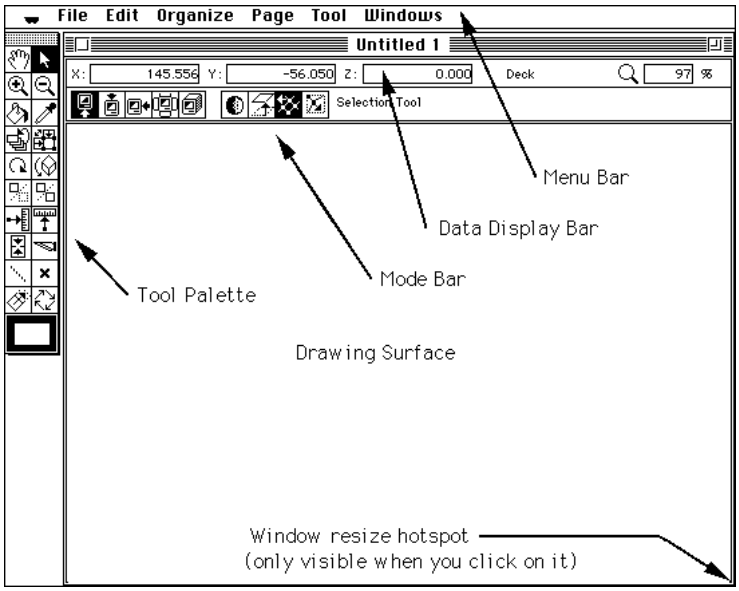
Coordinate system

The Touch-3D™ world coordinate system is organized as follows:



Drawing Window

When Touch-3D™ opens it will look as seen below.



Elements of the drawing window.

The Drawing window is divided into the following sections:

- The Title Bar where the file name is displayed.
- The Data Display bar which is used for displaying the values of the coordinates, and degree of magnification.

• The mode bar where views, drawing modes, sub tools and various messages are being displayed.

• The drawing area where models are drawn. Note that in the bottom right corner of the window there is a window resize hot spot, which is only visible when you hold the cursor over the resize area and press the mouse button.

Data Display Bar

The Data Display bar is located directly above the mode bar, and does two things. It displays the degree of zoom, and the present coordinates.

The coordinates are displayed in two modes:

• If no vertexes are selected, the Data Display bar reflects the coordinates of the cursor. If at least one vertex is selected, the Intersection point symbol occurs on the screen. The Data display bar then returns the location of the Intersection point, in the X, Y, Z directions. The Intersection point can be manipulated in a number of ways. The Data Display bar also returns a zoom degree value.

• The Data Display bar also allows you to type in new coordinates for the Intersection point directly from the keyboard. If you want to type in a new X coordinate, you either click on the X coordinate, or press the X key on the keyboard. You then type in a new coordinate value, and press the Return key.

Mode Bar

The Mode Bar contains buttons for views, modes and a message area.

Touch-3D™ has five standard views, Front, Top, Side, Unfold, and Perspective view. You can change view by clicking on any of these buttons.

The Mode bar also contains four selection mode buttons: Layer, Polygon, Mesh, Vertex, and Local mode. The Layer mode is used for selecting all polygons in a layer. The Polygon mode is used for selecting polygons as units, while maintaining the relative distance between the vertexes of each polygon. The Mesh mode is used for selecting vertexes and mesh points individually. The Vertex mode is used for selecting vertexes individually, one at the time. When you move and rotate vertexes in the Unfold view, you move all vertexes belonging to the selected component as well.

The Mode bar can, for some tools on the Tool palette, display various sub tool buttons. These buttons can either be used for selecting tool modes, or be used for initiating of a specific action.

The mode bar is also used for displaying messages from the program.

Drawing area.

The Drawing area is used for drawing and displaying models. In the bottom right corner there is a resize hotspot, allowing you to resize the drawing window. Note that you can't see it unless you press the mouse button while pointing within the hot spot area.

Tool Palette

The tool palette contains twenty drawing tools, and two tools for setting line and surface colors. Some tools display subtools in the Mode bar, which may be used for selecting modes, or to perform updating of the model. Some palette tools have alternative methods, which can be selected by double clicking on them.

Pan Tool

Pan Tool 

The Pan tool allows you to move the drawing around the drawing window like moving a sheet of paper on your desk.

Using the Pan tool.

To use the Pan tool, select it on the Tool palette, and drag the drawing around the screen. Note that the drawing also may be moved by clicking on one of the view icons. If a vertex is active, and displays the Intersection point, the coordinates of that vertex will be the located at the center of the drawing window, when the view is changed. This means that you won't "loose" the model somewhere in space where you can't see it. You can immediately identify the Intersection point, and continue working directly after having changed view. The Pan tool can be used in any view.

The Pan tool can also be accessed by pressing the Space bar. The Pan tool is then activated temporarily. To deactivate it and go back to the original tool, just press the Space bar again.

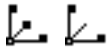
Selection Tool

Selection Tool 

The Selection Arrow is familiar to most Macintosh™ users. In Touch-3D™ the Selection tool is used for choosing items on the Tool palette, menus, commands and controls, dialog boxes, and for modifying elements in the drawing. The Selection tool can be used in any view for selecting objects, and for dragging and moving objects and vertexes. In the Unfold view objects are moved as units and not as individual vertexes.

Cross cursor. 

The Cross cursor is displayed when pointing within the drawing area, but not at a vertex.

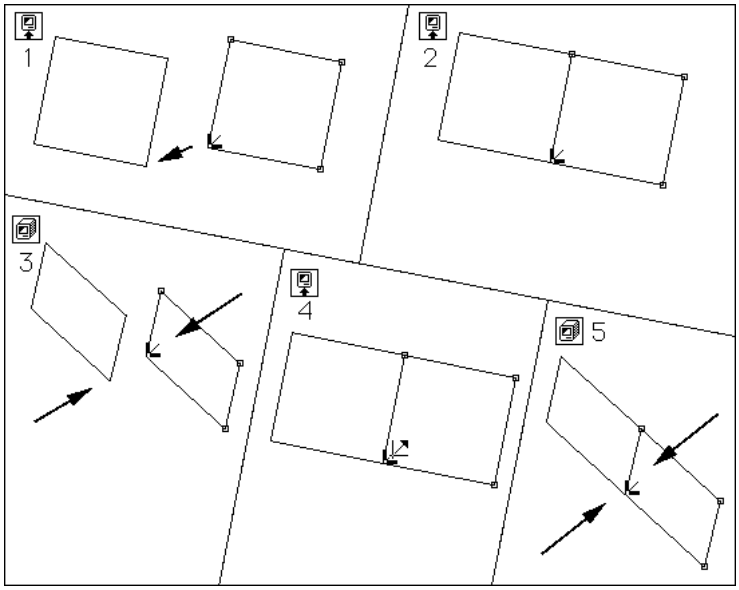
Coordinate cursors. 

Coordinate cursors are used for indicating a snap to a vertex, either in 2D or 3D.

When pointing at a vertex, the cursor changes into the Coordinate cursor. The appearance of this cursor varies depending upon the direction in which the cursor snaps to the vertex. The snap directions are indicated by dots at the end of the Coordinate cursor axes. When you click on a vertex, while displaying the Coordinate cursor, the Intersection Point symbol occurs on the screen.

Move 3D cursor. 

The Move 3D cursor is used for aligning the Intersection Point with an unselected vertex, in the direction away/towards you. Other selected vertexes are also moved a corresponding distance.



Using the Move 3D cursor.  -> 

Hold the cursor over the tip of the away/towards axes of the Intersection Point, as shown in the illustration above. The cursor changes into the Move 3D cursor. If you click when the Move 3D cursor is displayed, all selected vertexes move the same distance as the Intersection Point in the direction away/towards you. The moving distance is determined by the distance between the Intersection point and the closest aligning vertex, in the direction away/towards you.

The illustration above shows an example, where two polygons are intended to being connected. Note that step three and five only illustrates what happens. They are not a part of the method.

1/. The two polygons are not connected and not located at the same coordinate in the direction away/towards you. The polygons are seen in the Front view.

2/ Drag the left polygon to a location where the polygons overlap. Note that the cursor indicates a 2D snap, meaning that they align in the directions you see, but not in the direction you can't see (away/towards you).

3/ The difference can be seen in the Perspective view.

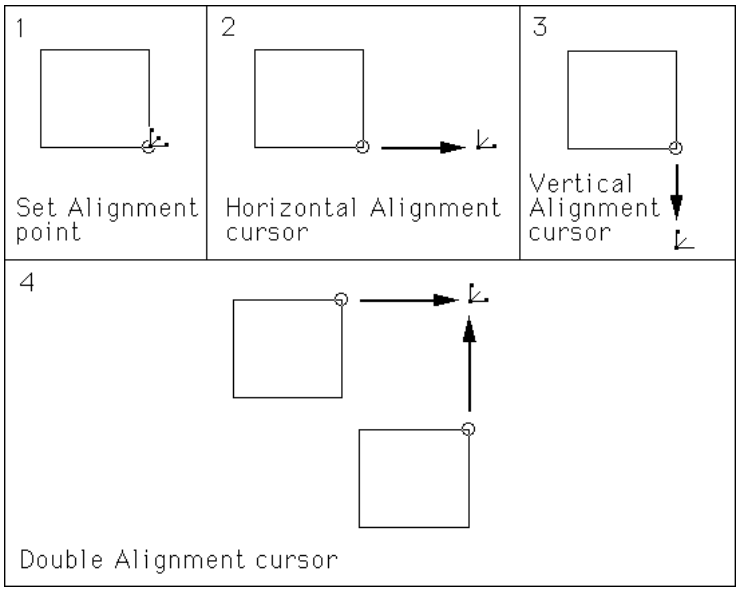
4/ Go back to the Front View. Move the cursor to a location over the tip of the away/towards you axis of the Intersection Point. Click when the Move 3D cursor occurs.

5/ When looking at the result in the Perspective view, the polygons now align in all three directions.

If there are several possible snap points, in the direction you can't see, the Move 3D function will select the snap point located closest to the Intersection point.

Alignment Indicator cursors.

The Alignment Indicator cursors are used for indicating alignments with vertexes.



Using Alignment Indicators.

When the Alignment Indicator function is turned on, the cursor can display the Vertical **(3)** or Horizontal **(2)** Alignment cursor, and a circle indicating the location of the aligning point. The Alignment Points (there are two) are selected by simply holding the cursor over a vertex **(1),** without clicking. A double alignment **(4)** is established by holding the cursor over a first vertex, and then a second vertex. When a double alignment occurs, the cursor displays dots at both the vertical and horizontal axes.

The Alignment Indicator is toggled on and off in the Preferences command found in the Page menu, or by pressing the C or c key.

Intersection Point. 

The Intersection point is used as a handle, for dragging selected vertexes to a new location.

The Intersection Point occurs when you click on a vertex. If more that one vertex is selected, other selected vertexes are displayed surrounded by red squares. The Intersection Point can be moved from one selected vertex to another, by clicking on the second vertex.

The coordinates of the Intersection Point are displayed in the Data Display bar, when at least one vertex is selected. If no vertexes are selected, the Data Display Bar shows the location of the cursor.

Dragging the Intersection Point.

The Intersection Point coordinates can be changed by dragging the Intersection Point on the screen. When the Intersection Point is moved, all selected vertexes will move correspondingly. In the Unfold view, all vertexes in an object/layer will be moved.

Entering new coordinates in the Data Display bar.

The Intersection point can also be moved by entering new coordinates in the Data Display bar. Coordinate values may be entered using three different methods.

The first method is used for entering absolute coordinates, where you simply type in the values for the coordinates concerned. The Intersection point then moves to the coordinate you just typed in, and all other selected vertexes move correspondingly, as if you drag the Intersection point on the screen.

The second method is used for relative movements, where you add an M or m before the coordinate to indicate a relative movement (E.g. 1000 -> M100 -> 1100). The value you type in is added to the present coordinate if it's a positive value, and subtracted if it's a negative value.

The third method is used for aligning all selected vertexes to the Intersection point. This is done by adding an equal sign (=) before the coordinate value (E.g. if you type in [=100] for a specific coordinate, all selected vertexes get the coordinate value 100).

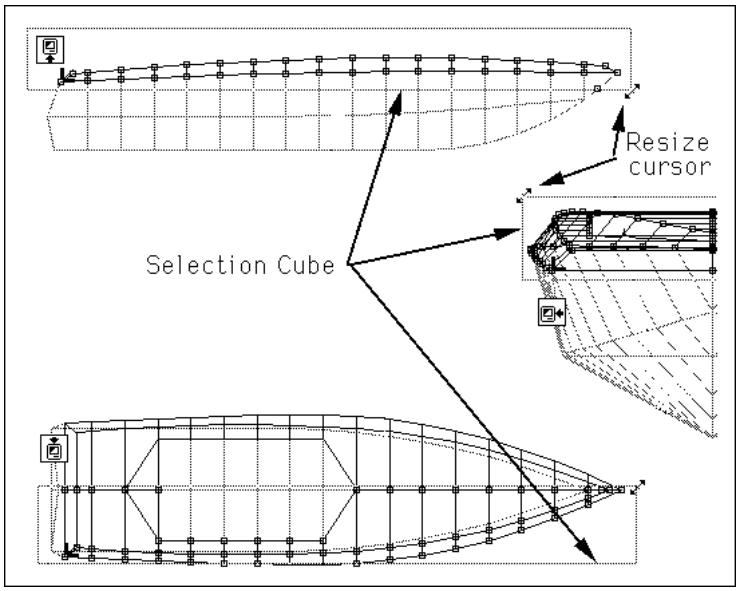
Moving the Intersection Point with the Arrow keys.

Another method of moving the Intersection point is to use the arrow keys. This applies to all views. The arrow keys behave according to view, which means that the Up key increases the Y coordinate in the Front and Side views, and the Z coordinate in the Top view. The Minus and Plus keys are used for movements away/towards you. The length of each step is determined by the settings in the Drawing Grid. The Grid is set by the Set Grid command in the Page menu.

Selection Cube & Resize Cursor.

The 3D Selection Cube is used for selecting more than one vertex at the time.

When you press the mouse button and drag the mouse, a selection cube is created. The Selection Cube remains visible until the vertexes are moved, and may be resized by holding the cursor over one of the Selection Cube corners. A Resize cursor occurs, and the box may be resized. When the mouse button is released, Touch-3D™ recalculates the selected number of vertexes, enclosed by the Selection Cube. Another very useful feature with the Selection Cube is that it's three dimensional. It may be resized individually in the Front, Top and Side views, thus creating a 3D Selection Cube. The illustration below shows the use of the 3D Selection Cube.



The behaviour of the Selection Cube changes with the Selection Mode.

Selection modes. 

When using the Selection tool, there are four behaviour modes, Layer, Polygon, Mesh, and Vertex Mode.

Layer Mode. 

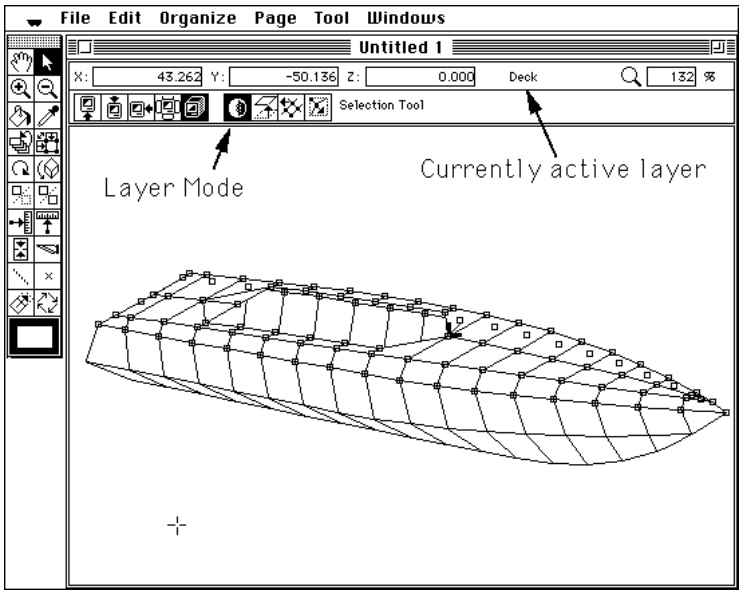
The Layer mode is used for selecting all polygons located in a specific layer.

Polygons can be selected by clicking on any vertex belonging to a layer. If the Layer Mode is set to Show / Snap / Modify Others (Layers menu -> Layer Visibility), polygons located in any visible layers may be selected. Otherwise, only the polygons in the currently active layer are selectable.

When a Selection Cube is used, all polygons of a layer have to be enclosed to be selected.

Using the Selection tool in Layer Mode.

In the illustration shown below, a boat hull is divided into two layers, the hull and the deck. When we click on a vertex on the deck, which is the currently active layer, all vertexes are selected.



Polygon Mode. 

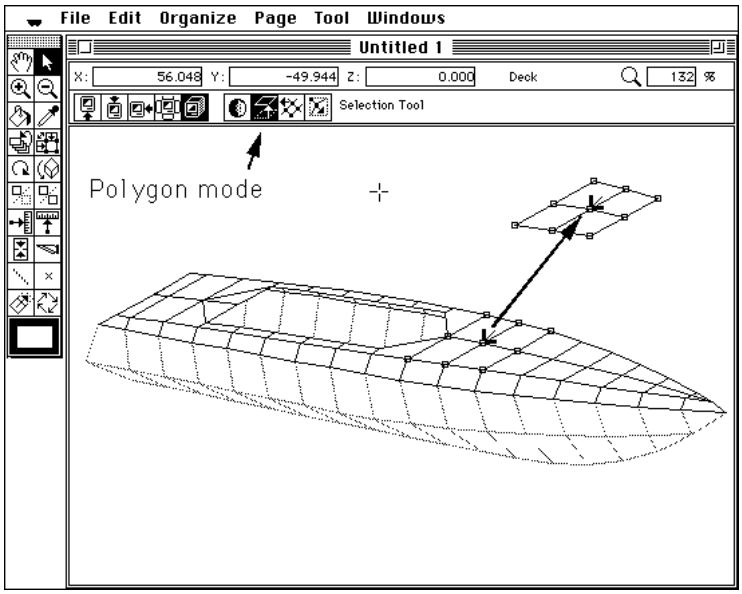
The Polygon mode is used for selecting polygons as units.

When you click on a vertex in Polygon Mode, all four vertexes in the polygon are selected. If the selected vertex is connected to vertexes in other polygons, the connecting polygons are also selected.

When a Selection Cube is used, only completely enclosed polygons will be selected. An individual polygon, within a mesh, may be selected by drawing a Selection Cube, enclosing that particular polygon. If a Selection Cube is drawn, while holding down the Option key, all polygons with at least one vertex enclosed by the Selection Cube will be selected.

Using the Selection tool in Polygon Mode.

The illustration below shows how some polygons in a model are moved. Click on a vertex, while being in the Polygon Mode, and drag the polygons away from the rest of the model. It's just as easy to drag it back to the original location. Click on a vertex, which will have a connecting vertex in the original model, drag it back to the original location, while carefully watching the cursor behaviour. When the cursor indicates a snap, release the mouse button, and click on the Move 3D point on the Intersection point symbol to snap in 3D (the hot spot is located at the tip of the away/towards you axis).



Mesh Mode. 

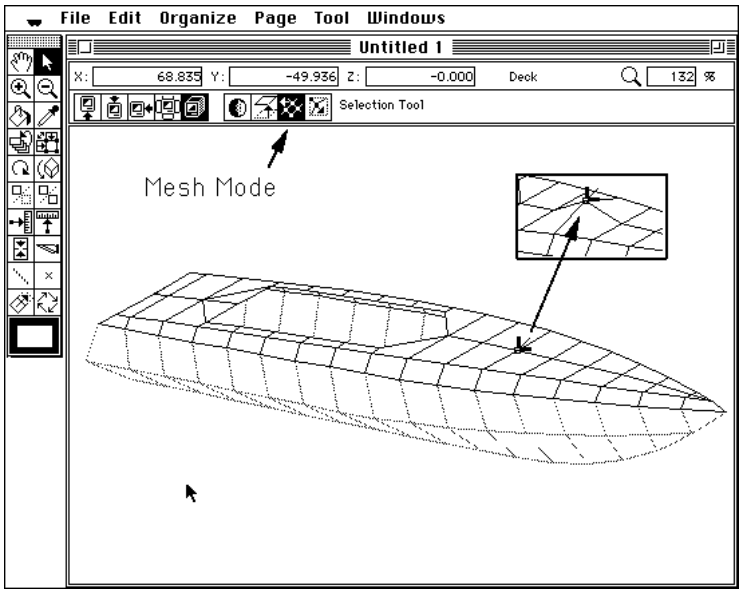
The Mesh Mode is used for selecting vertexes independently.

When several vertexes are connected to one another, all vertexes will be selected as a group. When a Selection Cube is drawn, all enclosed vertexes will be selected.

The Mesh mode is the default mode for Touch-3D™, and the most commonly used mode. It's mainly used for free form modelling and editing.

Using the Selection tool in Mesh Mode.

Below is an illustration showing how a vertex is selected, and dragged to a new location.



Vertex Mode. 

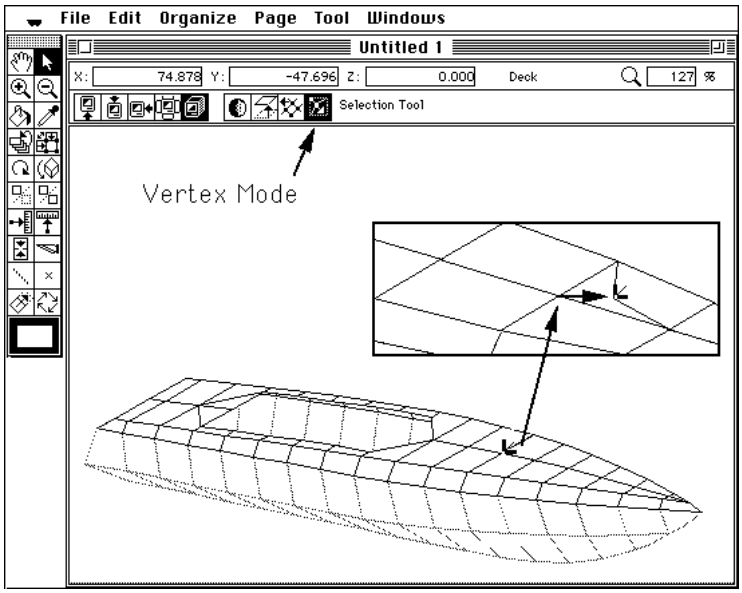
Vertex Mode is used for selecting vertexes individually.

When clicking on a vertex, where there are several connecting vertexes at the same location, only one vertex will be selected. If, instead, a Selection Cube is drawn, all enclosed vertexes will be selected.

The most common use for Vertex mode is to take apart a connection point. A good method to do this is to first place a Locus point at the location of the connection point. This enables you to move all vertexes, while saving the original location.

Using the Selection tool in Vertex Mode.

Below is an illustration showing how a vertex has been dragged away from the original location. Note that only one vertex is moved, while the rest of the vertexes remain at the original location.



Zoom Tools

Zoom in & out 

The Zoom tools are used for enlarging and reducing the view of your drawing.

You can zoom out to see the overall image of your drawing or zoom in to work on small details. A double click on the Zoom in / Zoom out tools will double / half the magnification.

The level of magnification is displayed in the mode bar. The value displayed is the level of change from normal scale, where the entire model(s) fits into the drawing surface. All reductions from normal scale are shown as values less than 100%, and enlargements are shown as values larger than 100%. If you click on the magnification level on the Mode bar, or press the "s" key on the keyboard, you will be able to enter a new value. Touch-3D™ will then zoom in /out accordingly and with the old center point as a new center point. The zoom value can range from 1% to 10000%.

You can easily access the opposite Zoom tool by pressing the Option key. Release it and the original tool setting is restored.

Note that you can set the magnification value to 100% by selecting Fit to Model Window (command / 4) in the Page menu.

Zoom in tool 

The Zoom-in tool allows you to select a specific portion of your drawing to be enlarged.

Using the Zoom In tool.

Select the Zoom in tool. You will now see the magnifying glass cursor with a plus sign in it. Place the cursor at the top left corner of the selected zoom area in the drawing area. Press the mouse button and drag the mouse to the bottom right corner. A finger cursor and a marquee will indicate the selected area. When you release the mouse, the selected area within the marquee will fill the screen. You may draw the marquee in any direction.

If you have selected the Zoom in tool and just click somewhere within the drawing area (no drag), Touch-3D™ will use that point as a new center point and simply doubles the magnification.

You can easily access the opposite Zoom tool by pressing the Option key. Release it and the original tool setting is restored.

The Zoom in tool can be used in any view.

Zoom Out tool 

The Zoom-Out tool allows you to select a specific portion of your drawing to be reduced in size.

Using the Zoom Out tool.

Select the Zoom out tool. You will now see the magnifying glass cursor with a minus sign in it. Place the cursor at the top left corner of the selected zoom area in the drawing area. Press the mouse button and drag the mouse to the bottom right corner. A finger cursor and a marquee will indicate the selected area. When you release the mouse, the selected area within the marquee will be the new center of the drawing area. If you draw a small marquee you will get a big reduction of the magnification and if you draw a marquee which is almost as big as the drawing area the reduction will be small. You may draw the marquee in any direction.

If you have selected the Zoom out tool and just click somewhere within the drawing area (no drag), Touch-3D™ will use that point as a new center point and simply half the magnification.

You can easily access the opposite Zoom tool by pressing the Option key. Release it and the original tool setting is restored.

The Zoom out tool can be used in all views.

Pick Up & Put Down Tools

The Pick Up and Put Down tools are a group of tools used for controlling the line and fill colors of the model.

Put Down Colors tool. 

The Put Down Colors Tool is used for changing fill and line colors of any selected polygons.

The Put Down Colors tool can be used in any view.

Put Down Color cursors. 

The Put Down Colors tool uses two cursors. They both look like a paint bucket, and differ only to indicate snapping or no snapping. When a snap occurs, a small dot is displayed at the top right corner of the cursor.

Using the Put Down Colors tool.

Selected the fill and line colors respectively. This can be done by clicking on the Fill Color or Line color tools at the bottom of the Tool palette. You can also pick up colors by using the Pick up Colors tool on the Tool palette. Select a suitable number of polygons (all vertexes of a polygon have to be selected). Select the Put Down Colors tool on the Tool palette. Move the mouse to a location over one of the selected vertexes. Check that a small dot occurs above the Put Down Colors cursor, and click when it does. The selected polygons will now change to the new color combination. Note that the fill colors are only visible when the Hidden lines command (Organize menu) is active. Note that the fill colors only appears when the Hidden Lines Removal command is activated.

Pick Up Colors tool. 

The Pick Up Colors Tool is used for picking up fill and line colors of a selected polygon. The color information of a selected polygon is displayed in Fill and Line Color tools at the bottom of the Tool Palette, when you hold the cursor over a selected vertex. Note that if you select several polygons with different color settings, the Pick Up Colors tool will display the color of the polygon you point at. The Pick Up Colors tool can be used in any view.

Pick Up Color cursors. 

The Pick Up Colors tool uses two cursors. They both look like a pipette, and differ only to indicate snapping or no snapping. When a snap occurs, a small dot is displayed at the bottom right corner of the cursor.

Using the Pick Up Colors tool.

Select a suitable number of polygons (all vertexes of a polygon have to be selected).

Select the Pick Up Colors tool on the Tool palette. Move the mouse to a location over one of the selected vertexes. Check that a small dot occurs above the Pick Up Colors cursor, and that the colors change to the right color combination in the Line And Fill Color tools on the Tool palette. Click to copy the colors. The default color setting is now changed.

Resorting tools

The resorting tools are two tools used for changing the order in which models are being unfolded.

Resort tool. 

The Resort tool is used for resorting the polygons of the model, to change the unfolding order. The Resort tool can be used in the Front, Top and Side views.

The Sorting Stack.

To understand what the Resort tool does, you have to know a little about how the unfolding function works.

When a first polygon is drawn (or imported into the drawing), it is placed at the bottom of a list of polygons. The list may be thought of as an imaginary stack of papers. When new polygons are added, the stack grows upwards, in the order in which they are drawn.

When the model is being unfolded, Touch-3D™ starts at the bottom of the stack, and combines surfaces into a bigger unit which forms the unfolded shape. If the model was created in an unstructured way, the unfolded model will end up being very unorganised, and may need lots of editing to be usable.

To sort the stack, the physical distance between the Resort Flag and each one of the selected polygons is being calculated. The polygon, located closest to the Flag, is placed at the bottom of the stack. The second closest polygon is next in the stack, and so on. The Resort Flag can be placed anywhere on the drawing surface, and can for this reason be used for changing the unfolding order in any way you like.

Resort Flag & cursor. 

The Resort Flag is used as a starting point for sorting. Polygons are sorted according to distance from the Resort Flag. Note that the flag is three dimensional, and that the sorting distances are calculated as 3D distances. For this reason, it's important to position the Resort Flag carefully in all three dimensions.

The Resort Flag is moved by dragging it with the mouse. When holding the cursor over the Resort Flag, the cursor changes into the Flag cursor. Press the mouse button and drag the Resort Flag to a new location.

Resort Execution buttons. 

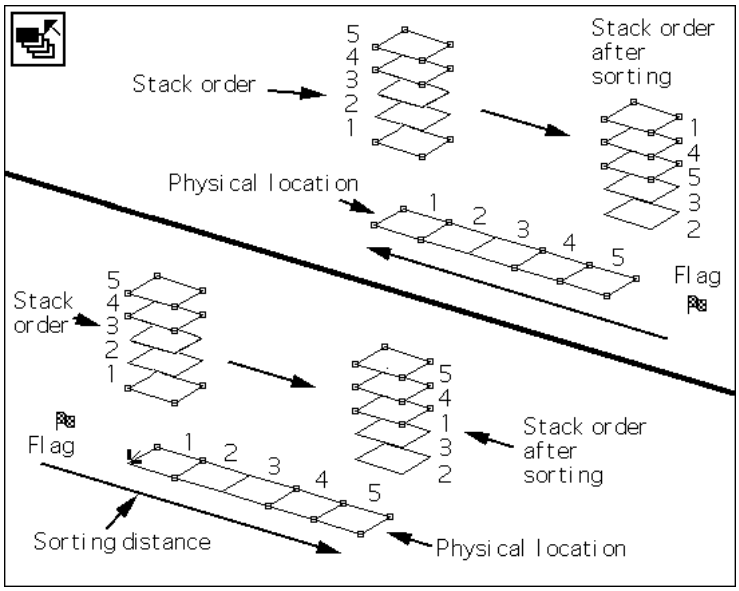
When the Resort tool is active, two resort execution buttons are displayed in the Mode bar. The purpose of these buttons is to initiate resorting of polygons.

Resort Upwards execution button. 

The Resort Upwards execution button, to the left in the Mode bar, places all selected polygons at the top of the stack, leaving the unselected polygons located below in the stack.

It then sorts the selected polygons according to distance from the Resort flag, with the polygon closest to the Resort Flag at the bottom, the second next to that, and so on. The result is a stack where the unselected polygons are all located below the selected ones.

The illustration below shows two examples of the Resort Upwards Execution button works. Note that the selected polygons always end up at the top of the stack, where they are sorted according to physical distance from the flag. The unselected polygons are placed below the selected ones, and remain unsorted.

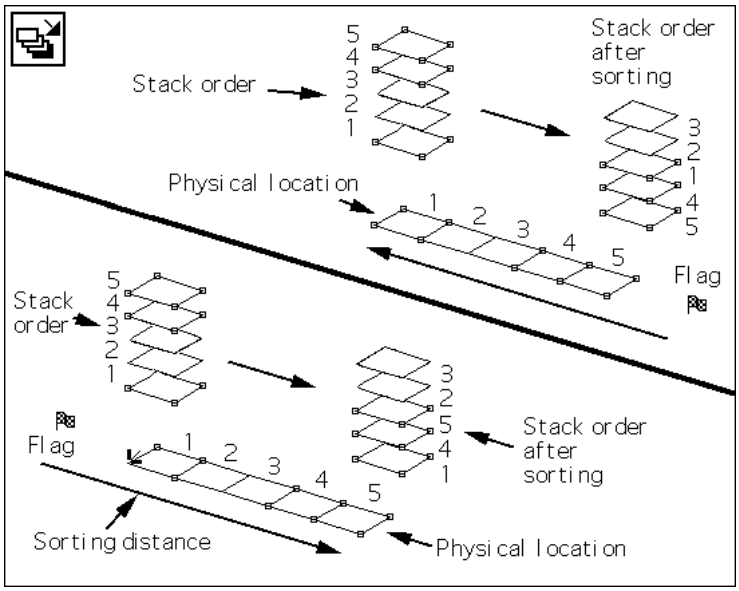


Resort Downwards execution button. 

The Resort Downwards execution button, to the right in the Mode bar, places all selected polygons at the bottom of the stack, leaving the unselected polygons located above in the stack.

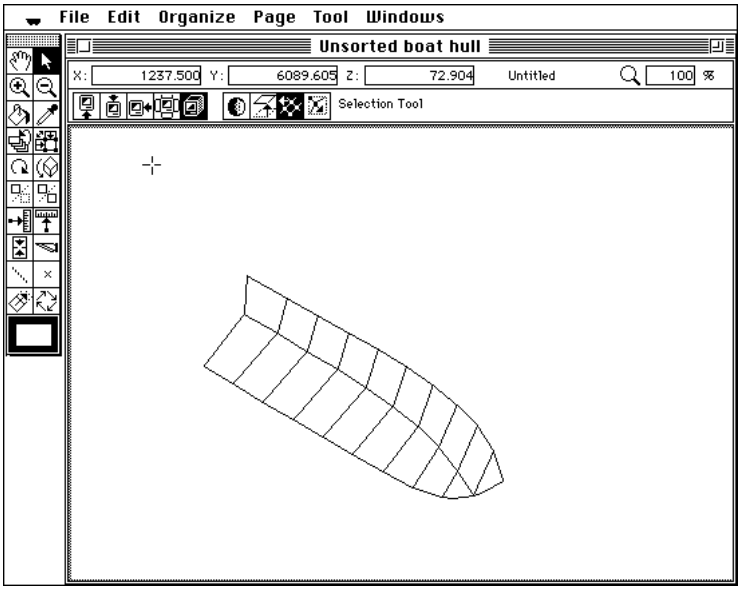
It then sorts the selected polygons according to distance from the Resort Flag, with the polygon closest to the Resort Flag at the bottom, the second next to that, and so on. The result is a stack where the unselected polygons are all located above the selected ones.

The illustration below shows two examples of how the Resort Downwards Execution button works. Note that the selected polygons always end up at the bottom of the stack, where they are sorted according to physical distance from the flag. The unselected polygons are placed above the selected ones, and remain unsorted.



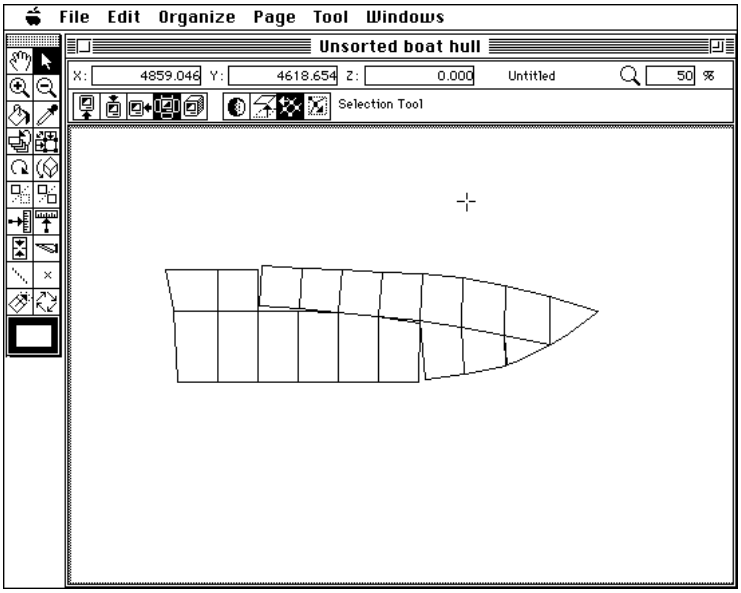
Using the Resort tool.

To resort a model, select a number of polygons (all corners in each polygon concerned have to be selected). Select the Resort tool. The Resort Flag occurs on the drawing surface (if not, try zooming out, it may be located outside the visible area). Drag the Resort Flag to a location, suitable for resorting the selected polygons. Click on either of the Resort Execution buttons to sort the polygons.

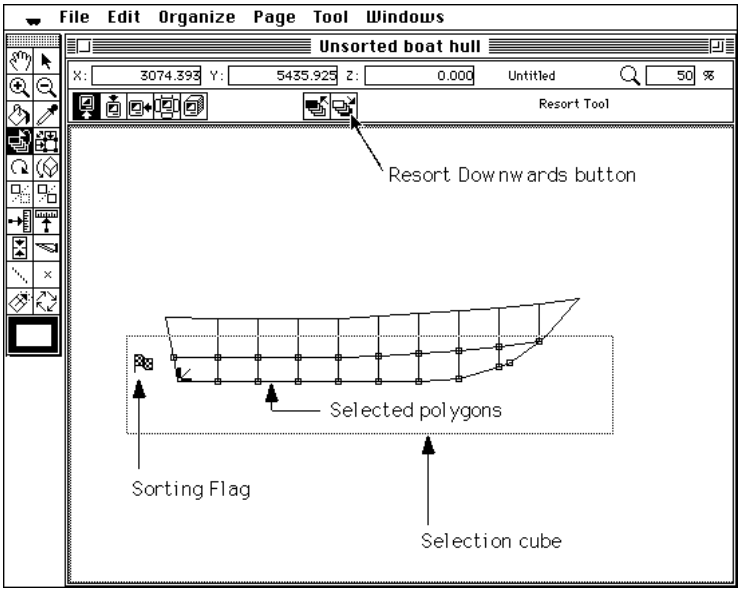


A practical example.

This is an example of how the unfolding order may be changed quickly. The unsorted model can be found in the Tutorial folder > Unsorted Boat Hull.

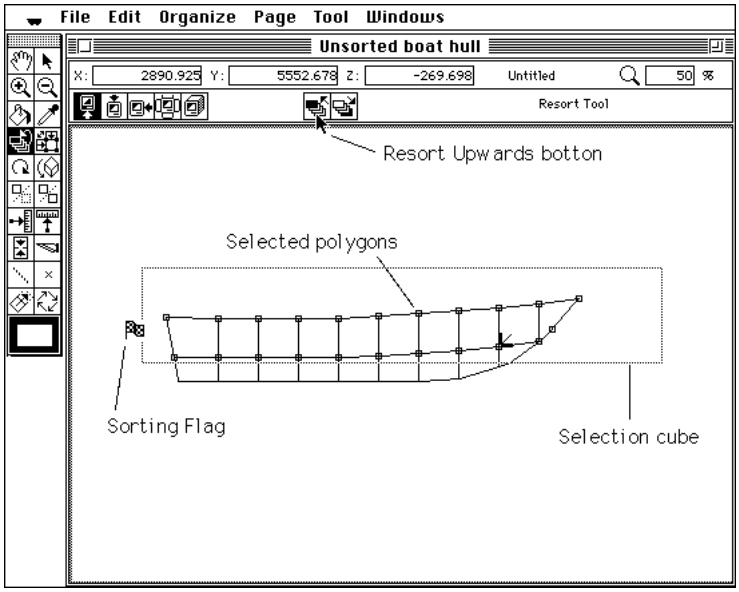


The illustration above shows that the model is very unsorted, as seen in the Unfold view. The bottom row represents the bottom of the hull, the upper row represents the side of the hull, and the bow is located to the right. Note that neither the bottom nor side panels are connected properly.

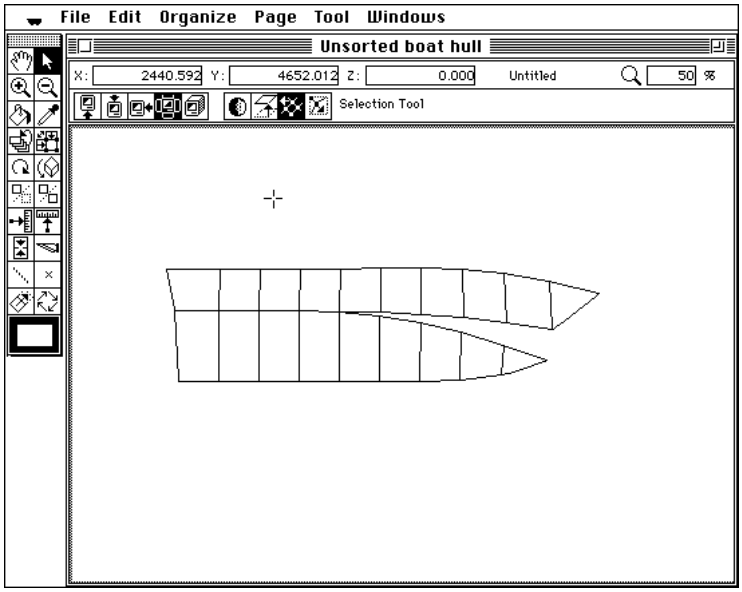


The intention is to resort the model in such a way that the bottom polygons are connected from left to right, and the upper (side) polygons are sorted from left to right, and with the top left polygon connected to the lower (bottom) row.

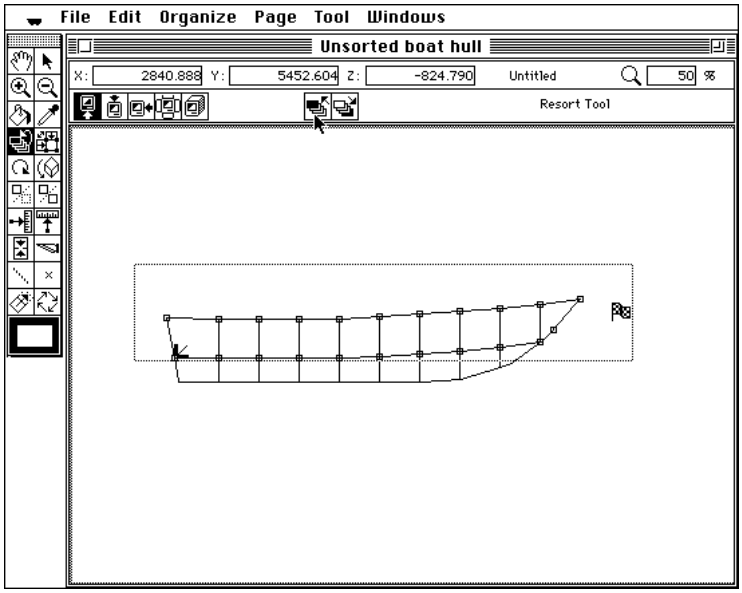
Select a number of polygons, (the bottom) by drawing a selection cube with the Selection tool. Select the Resort tool and drag the Resort Flag to a location to the left of the model, as shown above. Click on the Resort Downwards execution button in the Mode bar, to sort the selected polygons. The bottom left polygon will be placed at the bottom of the stack, because it's closest to the flag, followed by the rest in a row from left to right.



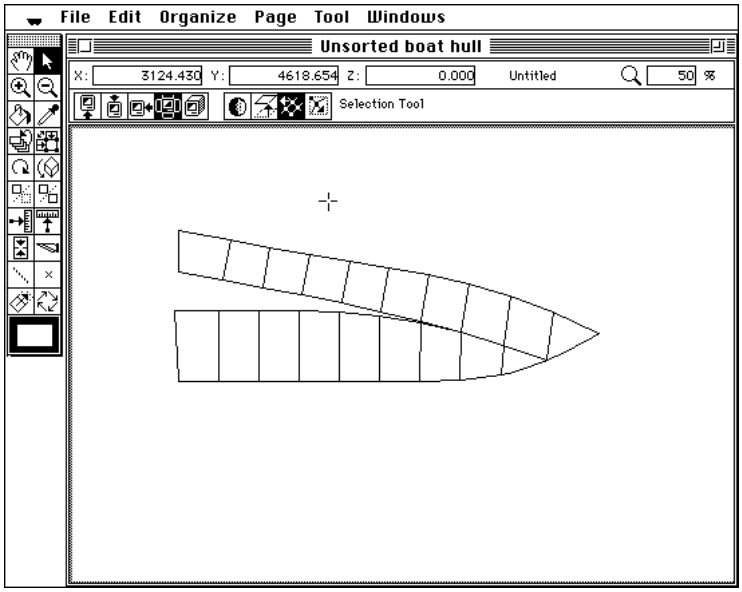
The next step is to resort the upper (side) row from left to right. The steps are the same as for the bottom. Select the upper row of polygons (the side), by drawing a selection cube with the Selection tool. Select the Resort tool and drag the Resort Flag to a location as shown above. Click on the Resort Upwards execution button in the Mode bar, to resort the selected polygons. The top left polygon will be placed immediately on top of the far right polygon of the bottom row, in the stack. The rest of the polygons in the upper row will follow from left to right in the stack. The reason we pressed the Resort upwards button instead of the Resort Downwards button is of course that we want to start building the model from the left end of the bottom row, and not the left end of the upper row.



To see the result unfolded, click on the Selection tool and then click on the Unfold View button. The model will now be unfolded as seen above, which is how it was intended to look like.



To sort the upper row from right to left, instead of the other way around, you follow the steps for the upper row as shown above, except for placing the Resort Flag to the right instead of to the left, as shown above.



The result will look as above.

Reconnect tool. 

The Reconnect tool is also a tool for resorting the unfolding order, but on a local level. When the Reconnect tool is used, only one polygon may be active at a time. The circles surrounding the selected polygon vertexes have two sizes, where the bigger ones indicate from which direction the model is being built.

The resorting is best done in the Unfold view, providing an almost instant update of an action, even though it works in any view. The tool is best used for fixing small errors, and should typically be used after a major resort has been done with the Resort tool.

Reconnect buttons. 

The Reconnect tool has three execution buttons located in the Mode bar, and are used for operating the tool.

Reconnect button. 

The Reconnect execution button is used for executing a reconnecting calculation for the presently selected polygon.

Note that the model may change appearance substantially when using the Reconnect execution button.

Another important thing to note is that the Reconnecting sequence works in a loop. This means that you will be able to go back to the original appearance if you click a number of times on the Reconnect execution button.

It's also important to understand that other polygons connected to the reconnected polygon may also be affected by a reconnection.

Step Down button. 

The Step Down execution button is used for stepping downwards, one step at the time in the stack. When you click on the button, the selection circles moves to a new location on the screen.

Step Up button. 

The Step Up execution button is used for stepping upwards, one step at the time in the stack. When you click on the button, the selection circles moves to a new location on the screen.

Using the Reconnect tool.

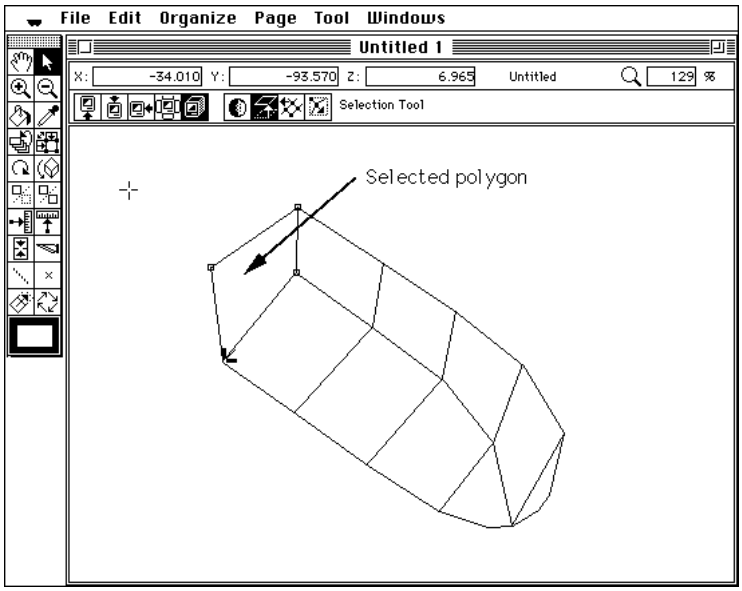
To use the Reconnect tool, click the Unfold view button in the Mode bar. Select the Reconnect tool on the Tool palette.

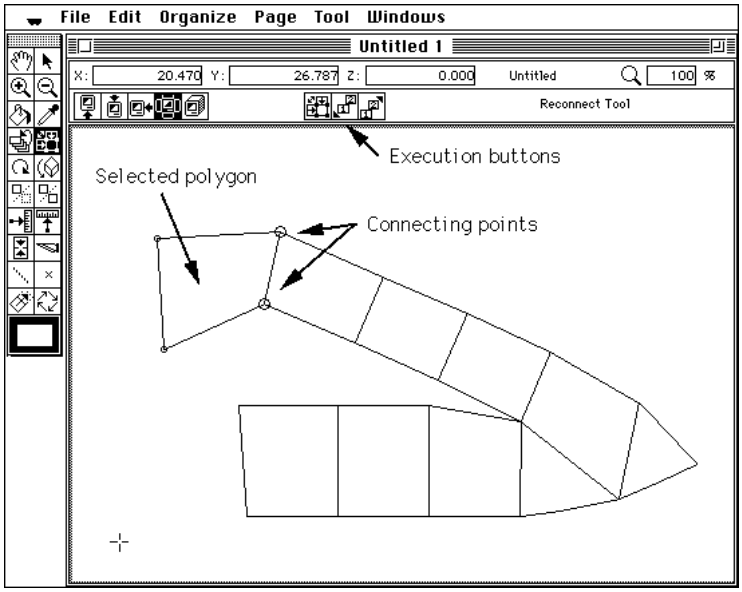
Use the Step Up or Step Down buttons to select the right polygon. If you select the right polygon before entering the Reconnect tool, this polygon will be pre-selected.

Click on the Reconnect execution button, until the model looks as you want it to look. Repeat the Step-Reconnect sequence with other polygons in the model if necessary.

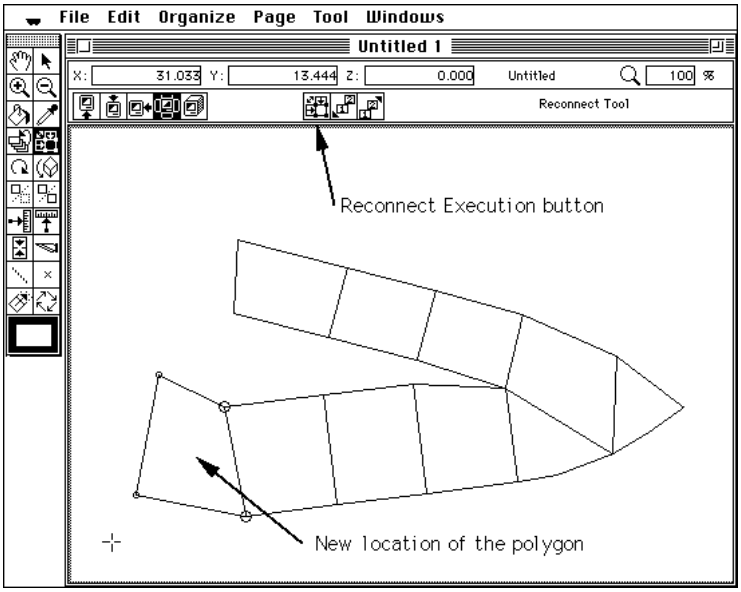
A Practical example.

Below is an example showing how to reconnect a polygon. The selected polygon has two possible connection directions, to the right, and downwards.





Click the Unfold view button (Mode bar). Select the Reconnect tool on the Tool palette. Use the Step Up or Step Down buttons to select the intended polygon. The illustration above shows how the model appears before clicking on the Reconnect execution button.



Click on the Reconnect execution button to modify the model. The illustration above shows the intended result .

Rotation tools

The Rotation tools are two different tools for rotating drawing elements and rotating the Perspective view.

Rotate Tool. 

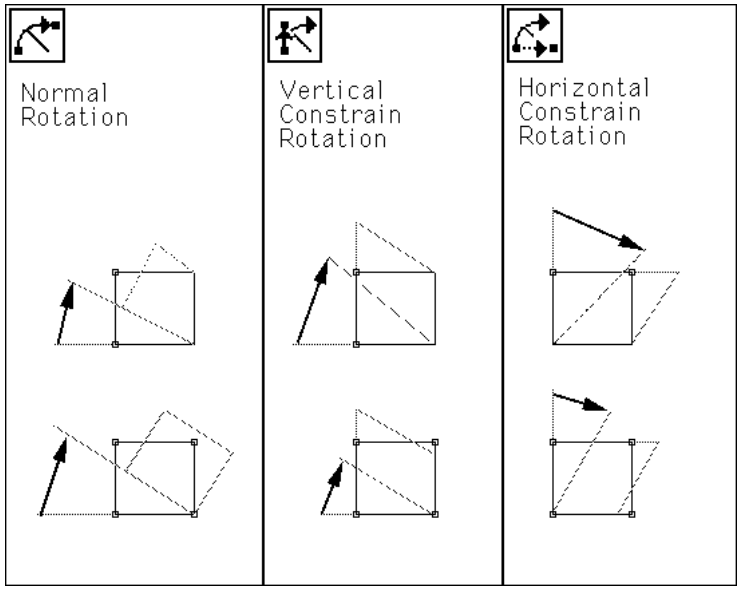
The rotate tool is used for rotating vertexes and/or polygons. The Rotate tool can be used in the Front, Top, Side, and Unfold views. When used in the Unfold view, all vertexes in an object or layer will be rotated, and you can only use the Normal rotation mode.

Rotation modes. 

When the Rotate tool is active, three rotation mode buttons occur in the Mode bar. The left button is used for normal rotation. The center button is used for constrained rotation, where only the vertical part of the rotating calculation is being performed. This results in a skewing effect in the up/down direction. The right hand button in the Mode bar does the same thing as the center button but horisontally. In the Unfold view, you can only use the Normal rotation mode.

Using the Rotate tool.

To use the Rotate tool, first select a number of polygons or vertexes. Select the Rotate tool. Click somewhere on the screen to place a center point of rotation. Click a second time at some other location to create a lever. All selected vertexes will now follow the lever. Drag the cursor to a suitable location and click to finish the sequence. To lock the lever in 0°, 45°,90°, 135°, 180°, 225°, 270°, or 315°, drag the mouse to a location roughly aligned with the intended angle, press the Shift key, and click to establish the second point.

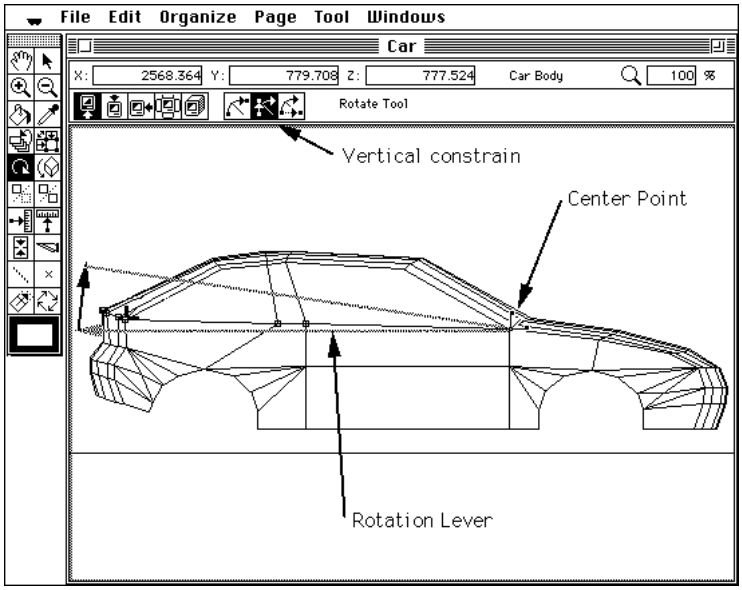


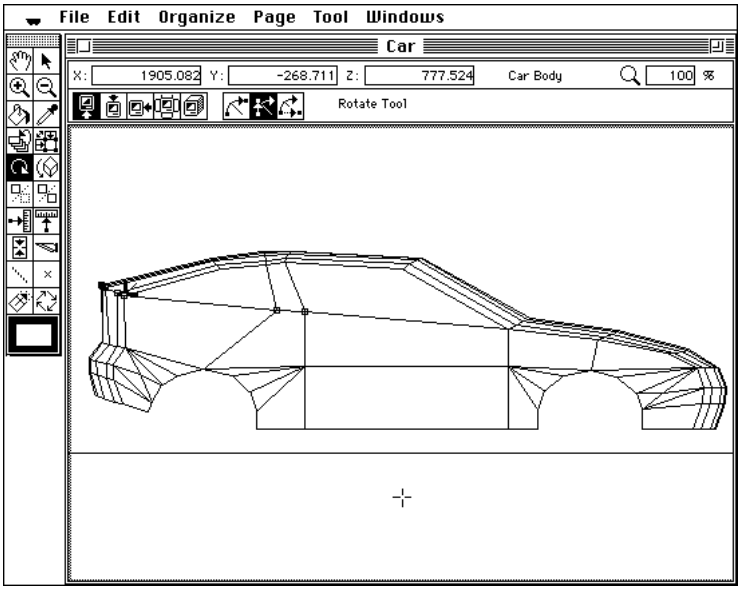
The illustration shows some examples of rotation using the three rotation modes.

Constrain Rotation example.

Below is a practical example of how the constrained rotation may be used.

All vertexes below the side windows, from the windshield and backwards have been selected. Click on the Rotate tool on the Tool palette, and select the Vertical Constrain mode in the mode bar. Draw a lever line, starting at the bottom back end of the windscreen, and pointing horisontally to the left, by clicking in both ends. Drag the lever upwards until you find a pleasing appearance, and click.





Rotate View tool. 

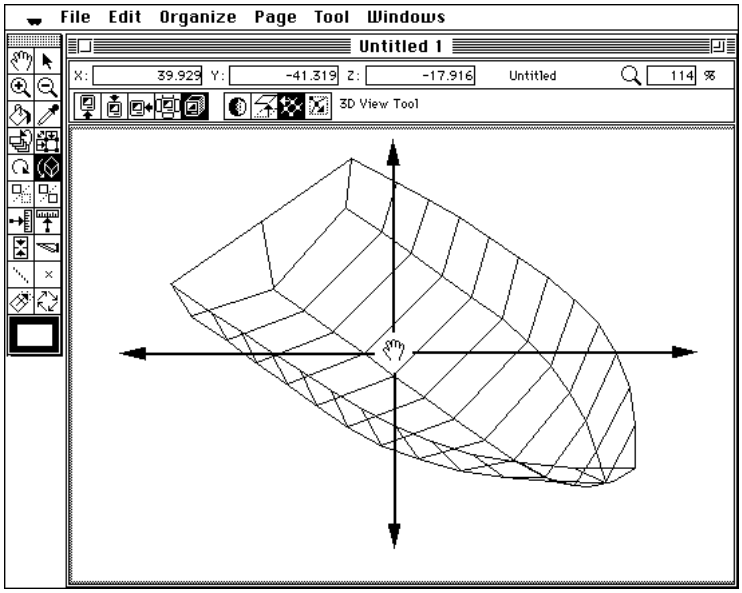
In the Perspective view, models can be seen from any 3D view. The Rotate View tool is used for rotating the view. Note that when you rotate the view, you do not change the location of the model, or rotate it. You only change the point from which you see the model. The Rotate View tool can only be used in the Perspective view.

Rotate Cursor.

The Rotate View tool displays the Hand cursor when you point within the drawing area.

Using the Rotate View tool.

To use the Rotate View tool, make sure you are in the Perspective view. Click on the Perspective view tool. Move the mouse to the center of the drawing area, as seen below. The Hand cursor appears. Press the mouse button, drag it slightly to the left, and release the mouse button. The model will be redrawn seen in a new view. Try rotating, using the same procedure as specified above, by dragging upwards, downwards and to the right, to feel how the Rotation View tool works.



Mirror Tools

The Mirror tools are used for mirror rotation and mirror copying of drawing elements.

Mirror tool. 

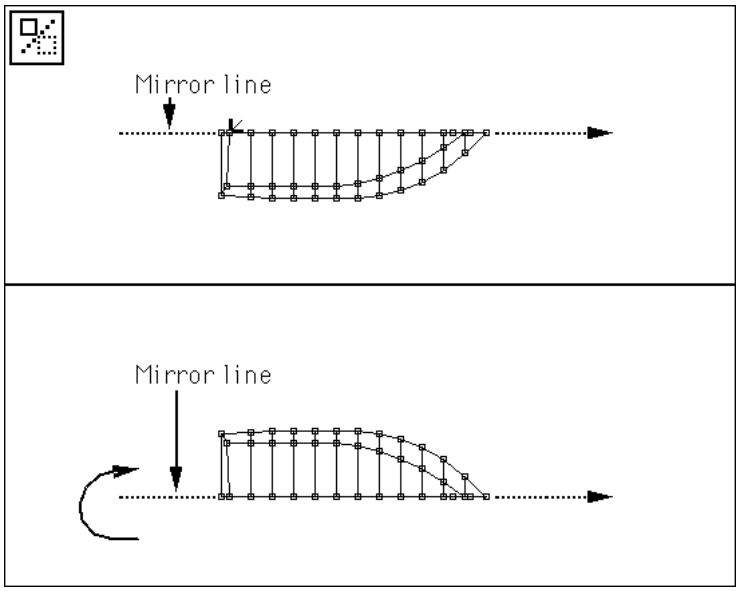
The Mirror tool is used for mirror rotating selected polygons or vertexes around a given axis.

The Mirror tool can be used in the Front, Top, and Side views.

Using the Mirror Tool.

To use the Mirror tool, select a number or polygons or vertexes, using the Selection tool. Select the Mirror tool on the Tool palette. Draw a mirror line around which the polygons will be rotated. When the line has been drawn, the selected polygons will be mirrored.

To lock the mirror rotation axes in 0°, 45°,90°, 135°, 180°, 225°, 270°, or 315°, click to establish the first point of the mirror axis, drag the mouse to a location roughly aligning with the intended angle, press down the Shift key, and click to establish the second point.



Mirror Copy tool. 

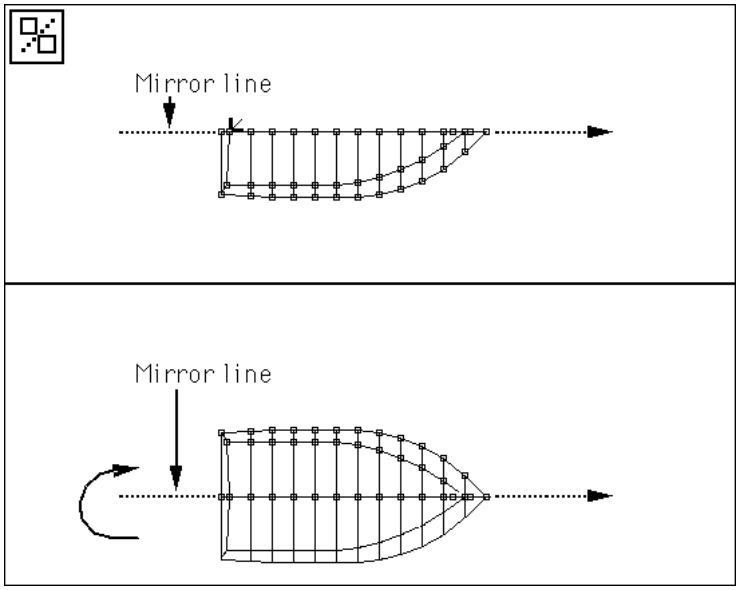
The Mirror Copy tool is used for mirror copying of a selected number of polygons around a given axis.

The Mirror Copy tool can be used in the Front, Top, and Side views.

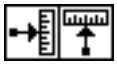
Using the Mirror Copy Tool.

To use the Mirror Copy tool, select a number or polygons, using the Selection tool. Note that all corners of each polygon have to be selected, since we are now copying polygons. Select the Mirror Copy tool on the Tool palette. Draw a mirror line around which the polygons will be mirrored and copied. When the mirroring line has been drawn, the selected polygons will be mirror copied.

To lock the mirror copying axes at 0°, 45°,90°, 135°, 180°, 225°, 270°, or 315°, click to establish the first point of the mirror axis, drag the mouse to a location roughly aligning with the intended angle, press the Shift key, and click to establish the second point of the mirror axis.



Alignment ruler tools

Alignment Ruler tools. 

The Alignment Rules tools are used for deforming any number of polygons in a wide range of ways.

One use for alignment rulers is for aligning a row of vertexes to a smooth spline curve, in 2D or 3D.

A second use is aligning a row of vertexes to a strait 3D line, in 2D or 3D.

A third use is deforming an entire model, or parts of it into an organic shape, in 2D or 3D.

The Alignment Ruler tools can be used in the Front, Top, and Side views. The rulers are also visible in the Perspective view, but may not be modified.

The Rulers are can be printed out on a printer if they are visible on the screen.

The Rulers can be made invisible by using the Hide Ruler Command in the Tool menu.

Ruler Cursors. 

The Ruler tools use one special cursor for indicating a snap to a control point. It is displayed when you hold the cursor over a Ruler control point.

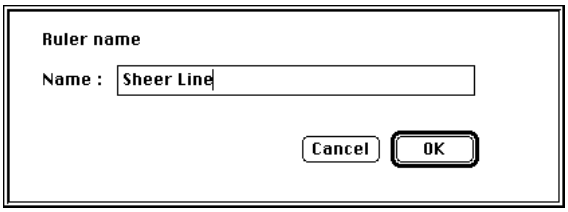
Creating rulers and deleting rulers.

Select either the Vertical or Horizontal alignment Ruler tool on the Tool palette.

If there are no previous rulers in the drawing, there are two possible methods available when creating a new ruler.

In the first method you create a ruler by clicking out ruler control points on the drawing surface. In this case a new ruler, called "Untitled--F,-T,-S", will be created. The ruler may be renamed later on, using the Rename Ruler command, on the Tool palette.

In the second method, you create a ruler by selecting the command New Ruler on the Tool palette, or by double-clicking on one of the Ruler tools. A dialog box occurs, asking you to name the ruler.

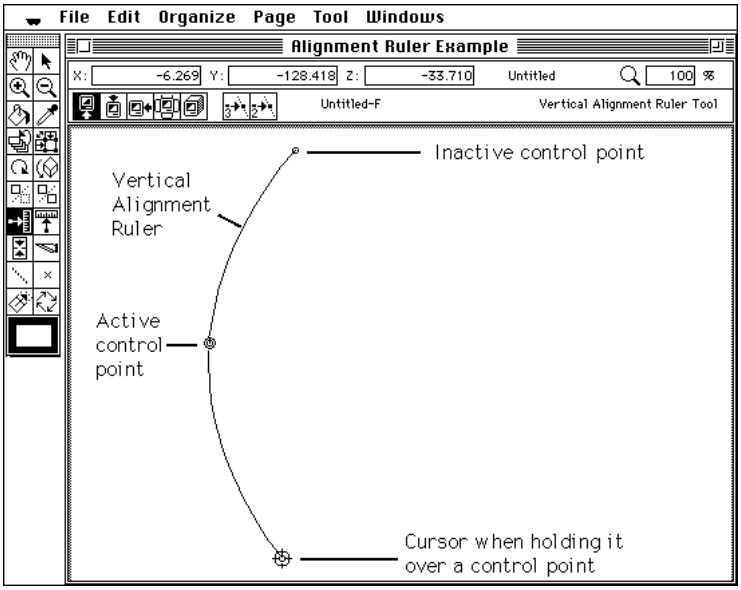


If there is at least one ruler, already created in the same main direction, you have to use the second method, because you would otherwise add new control points to the old ruler rather than creating a new one.

To delete a ruler from the drawing, double-click on any of the control points. When all points are selected, press the Delete key to delete the ruler.

Copy and paste rulers.

Rulers can be copied and pasted between different Touch-3D™ drawings, using standard Macintosh Copy and Paste commands.



Creating a smooth curve.

Ruler points are added to a ruler by clicking on the screen within the drawing area.

All rulers have a main direction, either horizontal or vertical. The reason for this is that it's impossible to calculate an alignment if you have more than one possible alignment point.

When a new control point is added, it is placed in the database of the program according to its physical location in the drawing. This also applies if you move a control point. When two control points have been added, a strait ruler is being formed. The strait ruler is used for aligning vertexes in a strait line. If the ruler contains at least three control points, a smooth spline curve is being created. The curve is used for aligning vertex points to the curved ruler.

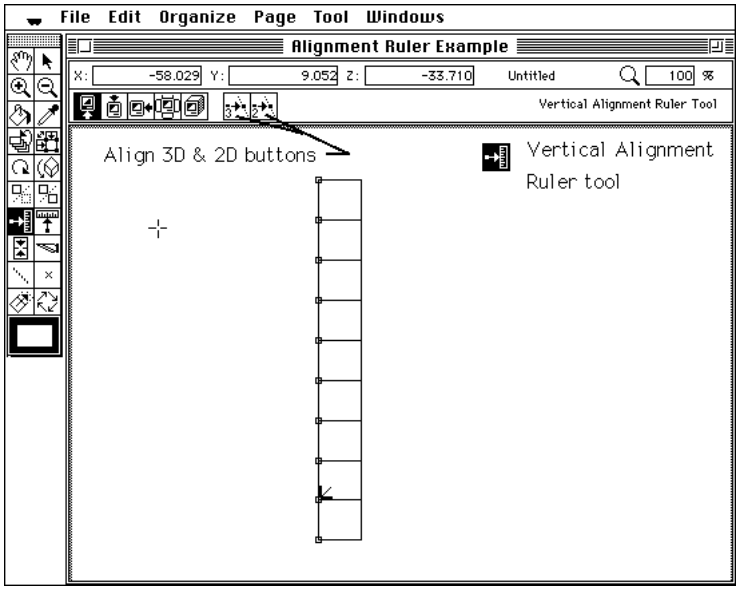
When clicking on a vertex, the control point snaps to the vertex. To move a control point you click on it to select it. It will change from green to red and grow in size. The control point may then be dragged to a new location. To lock the dragging horisontally, vertically, or diagonally, you press the Shift key. Several control points

may be moved simultaneously by Shift-clicking on more than one control point. All control point may be moved by double-clicking on any of the control points. A control point may be deleted by clicking on it and pressing the Delete key. A ruler can be deleted completely by double-clicking on a control point, and pressing the Delete key. A control point can be moved by using the arrow keys if it has been selected. This is useful particularly for small movements.

A practical example.

Below is an example of how Alignment ruler may be used.

First draw some polygons. Select the left row of vertexes using the Selection tool (Mesh mode). Click on the Vertical Alignment Ruler tool to select it. Note that two buttons occur in the mode bar.

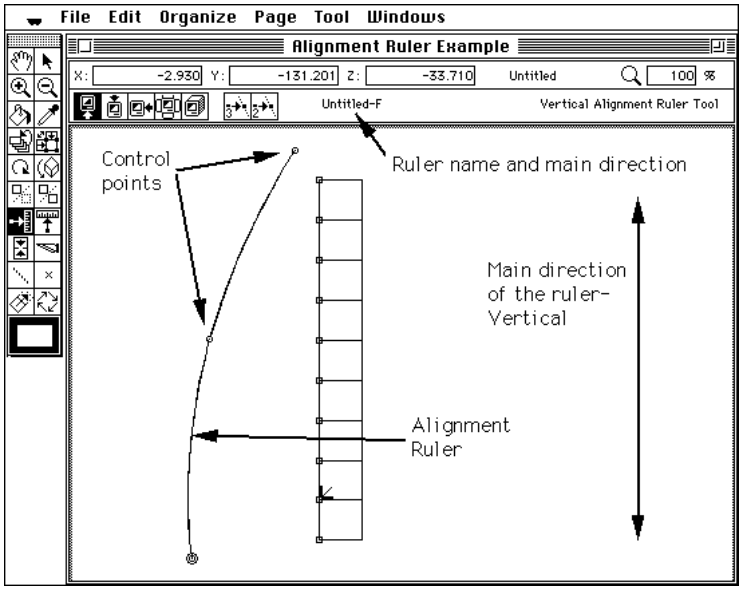


Draw a ruler to the left of the polygons by clicking out three control points as seen below. Inactive control points are displayed on the screen as green circles, and active points are displayed as two slightly bigger red circles.

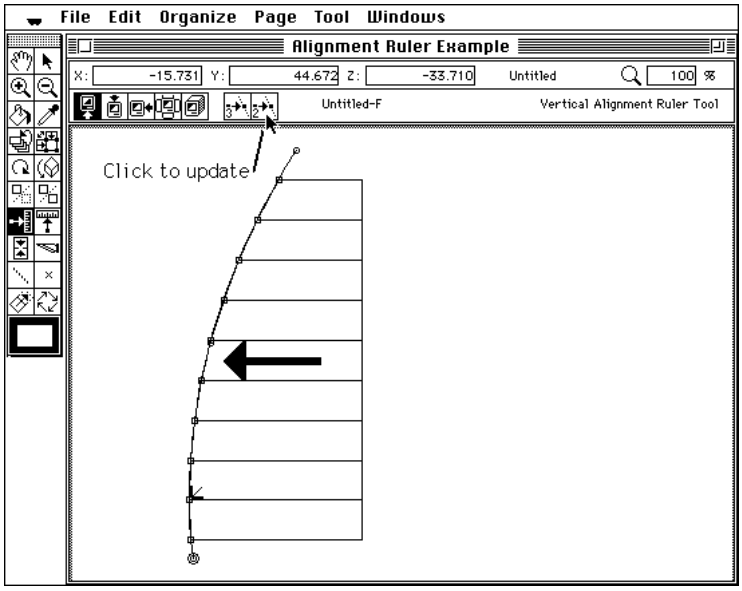
Note that the ruler is given a name, and a main creation view, in which it was created. The reason for giving the ruler a name is that you may use more than one ruler in a drawing, and in that you at any time may change active ruler. Rulers may be renamed by using the Rename Ruler command found in the Tool menu.

The (F) after the ruler name indicates that it was created in the Front view. Since the ruler was created in the Front view as a vertical ruler, there are some restrictions how it can be modified. Control points can only be added in the Front and Side views, since the main ruler direction has to be seen, when adding points. This is not the case in the Top view, because you can't see the up/down direction. Control points may, however, be moved and deleted in either the Front, Top or Side views.

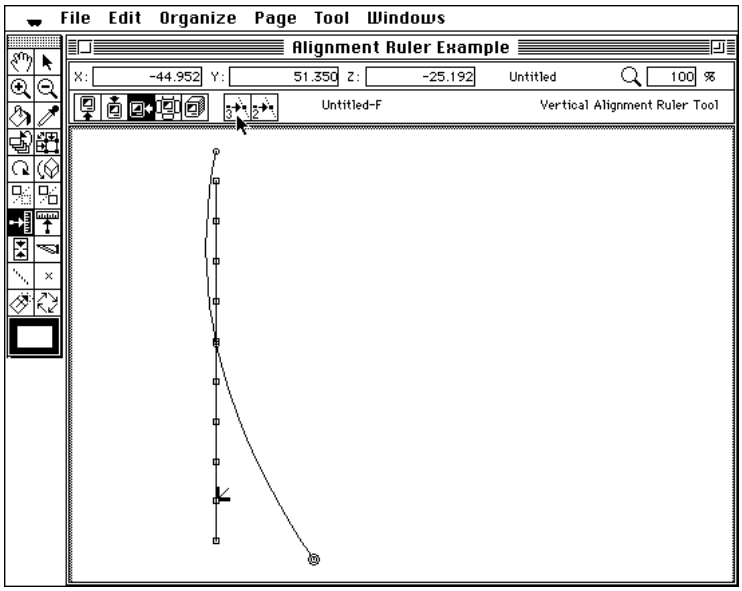
It's, therefore, important to know if a ruler is vertical or horizontal, and in which view it has been created.



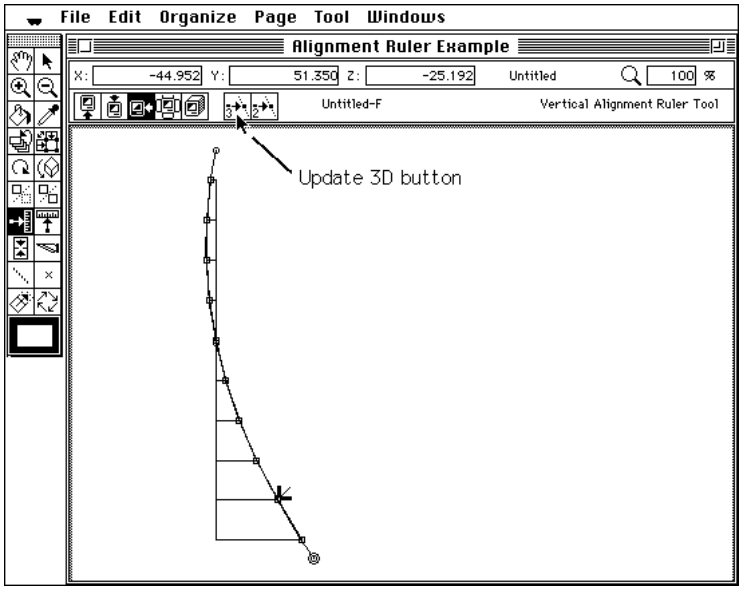
To align the selected vertexes to the ruler, you simply click on either the 2D or 3D align buttons found in the Mode bar. The difference is that the 2D button only moves the vertexes as seen in the current view, whereas the 3D button updates the vertexes in all three directions. The illustration below shows what happens.



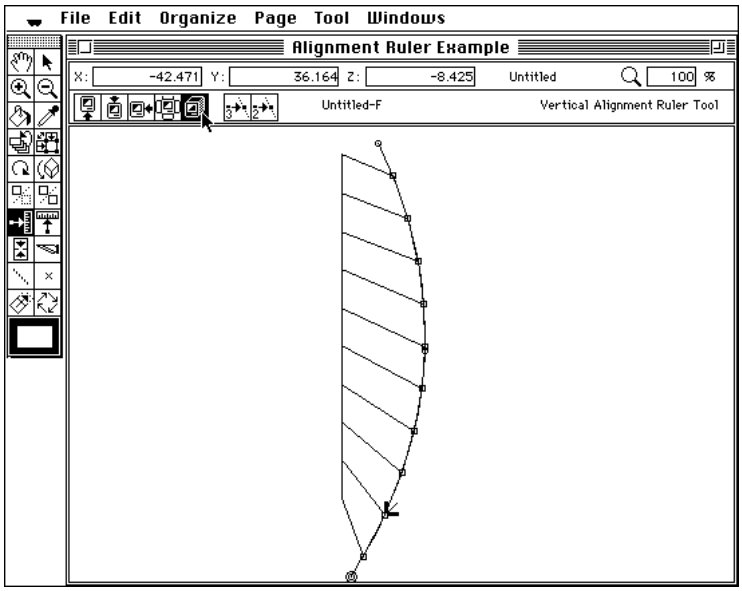
So far, we haven't used the ruler in 3D. To do that, simply change view to a view where the directions not seen in the Front view can be seen, for example the side view. Drag the ruler control points so that it forms a curve, as seen below.



Click on the Update 3D button in the Mode bar to update the model. The model will appear as below.



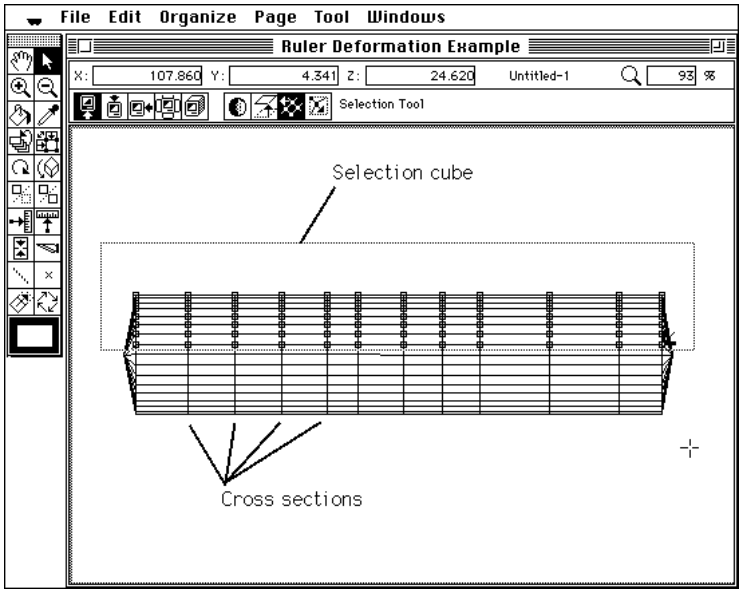
In Perspective view it will look as below. Note that the rulers still are visible, but not editable in this view. To hide the rulers, select the Hide Rulers command in the Tools menu.



Deforming a model.

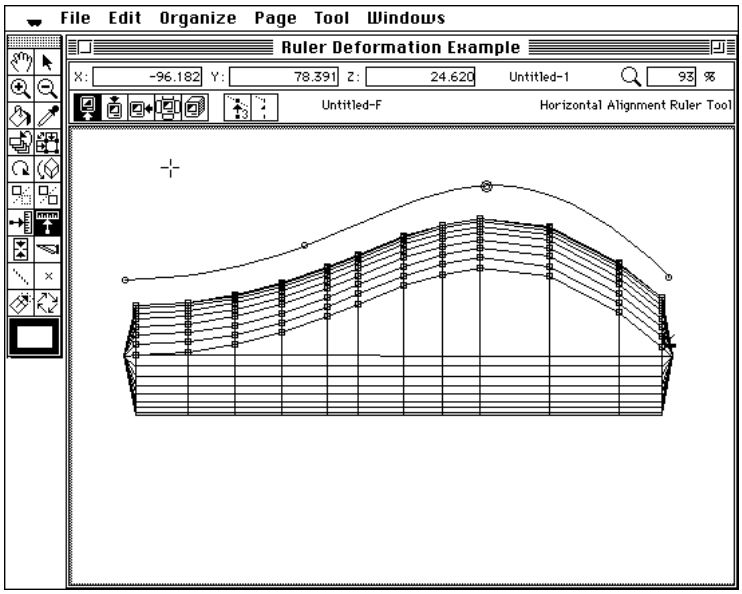
Another way of using Rulers is to deform models. Below is an example of how it can be done.

Create for example a horizontally oriented cylinder as model. To do this, import a shape called Cylinder 36 (indicating that the circular shape is divided in 36 steps) from the Shape folder. Rotate the cylinder from vertical to horizontal. Stretch it to make it longer. Divide the cylinder into a number of sections, using the Cut tool found on the Tool palette. Select the upper half of the model, using the Selection tool in Mesh mode. The model now looks as below.

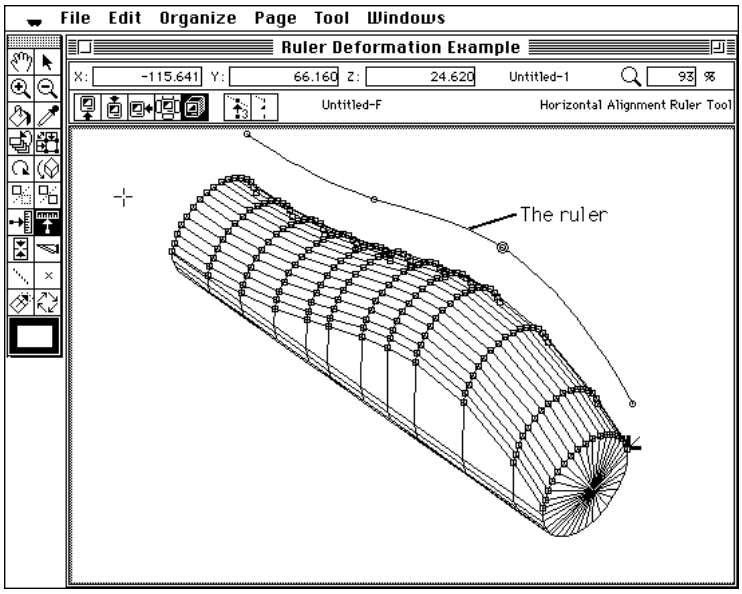


Select the Horizontal Alignment Ruler tool on the Tool palette. Draw a ruler, located slightly above the cylinder. The reason for locating the ruler above the cylinder is that it's easier see it, and less cluttered. Before starting to deform the cylinder, we have to lock the selected vertexes to the ruler movements. This is made by selecting the Lock To Ruler command in the Tool menu. This is a function that may be toggled on and off at any time. When this has been done, click on one of the ruler control points to select it, and drag it around a bit. Note that the selected vertexes now follow the movements of the ruler.

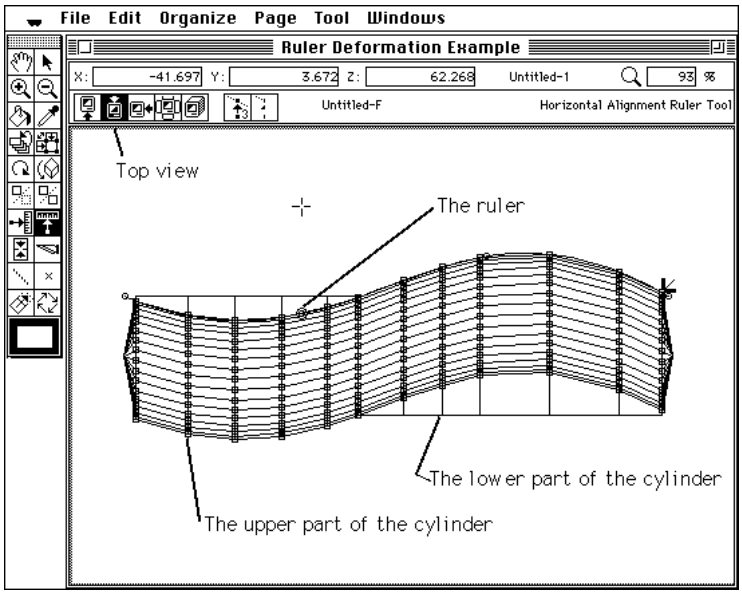
A deformed model may look as seen below.



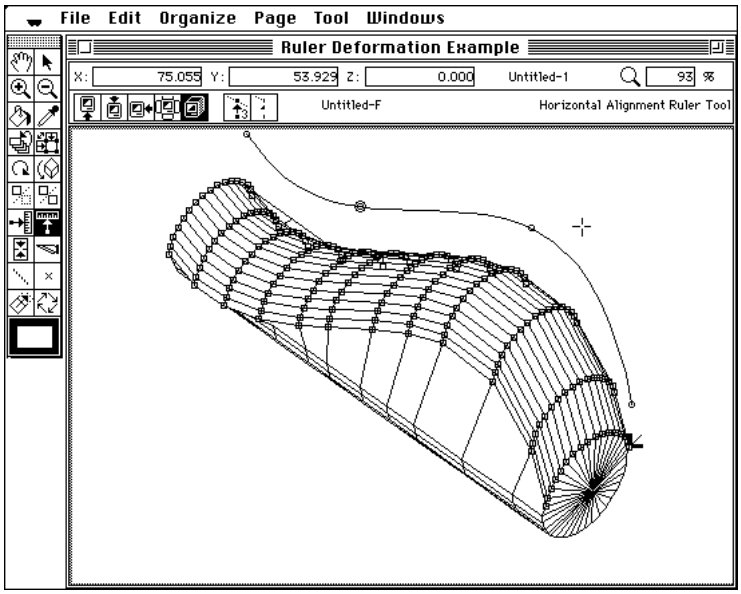
When looking at the model in the Perspective view it looks as seen below. Note that the ruler also is visible, even though it can not be used in this view.



To increase the complexity of the model, move to the Top view by clicking on the Top view button in the Mode bar. Then, deform the model in the Top view, creating, for example, a wave form. It may look like the illustration given below.



In the Perspective view, the model may look as below.



Vertical Alignment Ruler tool. 

The Vertical Alignment Ruler tool is used for aligning vertexes horizontally to a ruler, or for deforming models horizontally. The direction is always defined in the view in which the ruler was first created.

The Vertical Alignment Ruler tools can be used in the Front, Top, and Side views. The rulers are also visible in the Perspective view, but may not be modified.

Rulers can be printed out on a printer, if they are visible on the screen. Rulers can be temporarily hidden using the Hide Ruler Command in the Tool menu.

Horizontal Alignment Ruler tool. 

The Horizontal Alignment Ruler tool is used for aligning vertexes vertically to a ruler, or for deforming models vertically. The direction is always defined in the view in which the ruler was first created.

The Horizontal Alignment Ruler tools can be used in the Front, Top, and Side views. The rulers are also visible in the Perspective view, but may not be modified.

Rulers can be printed out on a printer, if they are visible on the screen. Rulers can be temporarily hidden using the Hide Ruler Command in the Tool menu.

Join & Cut Tools

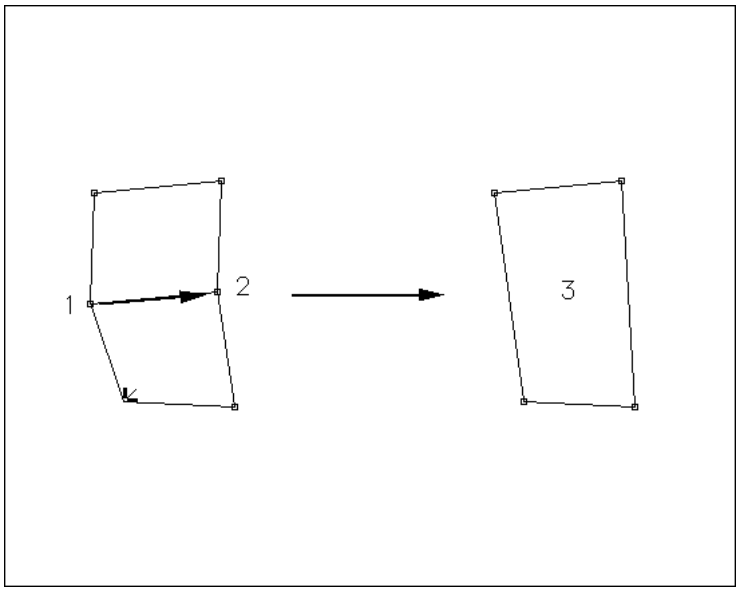
The Join and Cut tools are used for joining and cutting polygons.

Join tool. 

The Join tool is used for joining two polygons into one.

Using the Join Tool.

To use the tool, first select at least two polygons, having connecting sides. All four corners have to be selected in both polygons. Select the Join tool on the Tool palette. Click on the first vertex where the two polygons are connected (1). Click on the second vertex where the two polygons are connected (2). The two polygons will merge into one polygon (3). The Join tool works in all views, including the Perspective and Unfold views.



Cut tool. 

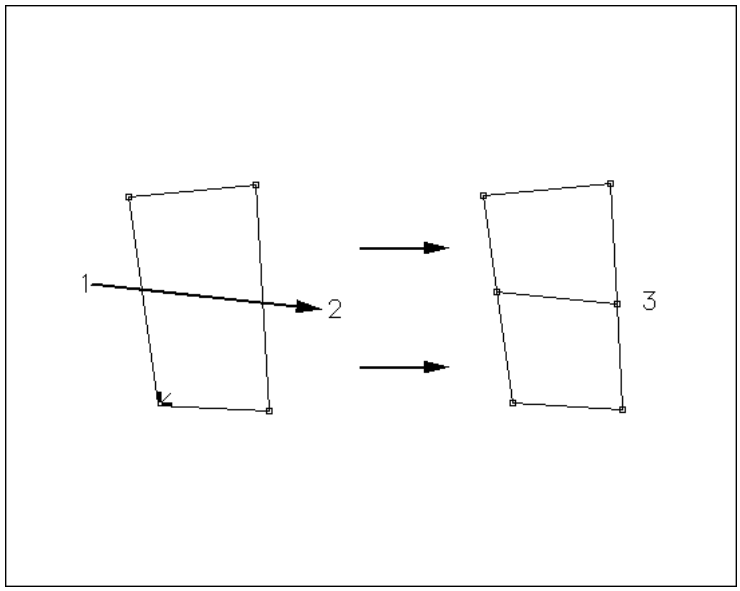
The Cut tool is used for dividing selected polygons into smaller polygons.

The Cut tool provides a significant design function, because it allows you to start building a model at a comparatively rough level using few polygons, and then gradually improve the resolution (number of polygons) at locations where required.

In some cases, the Cut tool divides a polygon into more than two polygons. This happens when the cutting results in a polygon having more than four vertexes. In such a case, Touch-3D™ divides the polygon into smaller units.

Using the Cut tool.

To use the Cut tool, select a number of polygons using the Selection tool. The Cut tool can cut any number of polygons at any time. Note that only polygons with all four corners selected will be cut. Select the Cut tool on the Tool palette. Draw a cutting line through the polygon(s) you want to divide(1-2). Note that the cutting line has to cross at least two sides of a polygon, or connect to a vertex of a polygon. The cutting line will divide all polygons it crosses (in a valid way) (3). In the example below, the polygon to the left is divided into two separate polygons. The Cut Tool works in all views, including the Perspective and Unfold views.



Help line & Locus tools

The Help Line and Locus tools are used for creating help and reference points, and are used as a modelling aid.

Help Line tool. 

The Help Line tool is used for creating 2D help lines.

Help lines are strictly two dimensional, and are linked to the view in which they were created. A help line, for example created in the Front view, can only be seen in the Front view, and so on. The Help Line tool may only be used in the Front, Top, and Side views. Help lines can only be modified when the Help Line tool is activated.

Help lines can be printed out on a printer if they are visible on the screen. To hide help lines, select the Hide Rulers command in the Tool menu.

Help lines are not exported when exporting models to external formats.

Help lines may, for example, be used for creating temporary snap points, required when drawing 3D polygons. Another example is to locate an intersection point between two lines, using the Join Help Lines command in the Tool menu. A third example is to draw a help line, with exact coordinates, to provide an accurate positioning when using the Cut tool. A fourth example is to provide simple drawing elements, for example showing the location of cross sections, or base lines.

Using the Help Line tool.

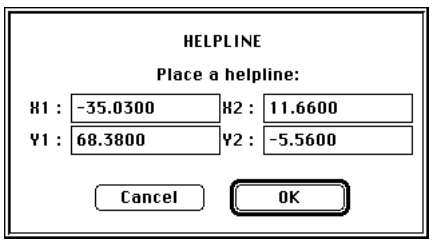
To use the Help Line tool, select the Help Line tool on the Tool palette, and click out the line on the drawing.

Drawing Help Lines.

There are several ways to draw a help line:

In the first method, simply click on the screen twice for the start and end points of the line.

In the second method, double click on the Help Line tool. The Help Line dialog box occurs allowing you to type the coordinates for the start and stop point of the line.



In the third method, select the New Help Line command in the Tool menu. The Help Line dialog box occurs, allowing you to type the coordinates for the start and stop point of the line.

Selecting help lines.

Several help lines may be selected with the Help Line tool, by drawing a selection box. All help lines completely enclosed by the selection box will be selected. Another method of selecting several help lines or help line vertexes is to shift click on each of them.

Deleting help lines.

Help lines may be deleted by selecting at least one of the vertexes in a help line and press the Delete key. All help lines in a drawing may be deleted by using the Clear All Help Lines command in the Tool menu.

Reshaping Help lines.

Help lines can be reshaped using one of the following methods after having selected the Help Line tool on the Tool palette:

In the first method, click on a vertex of a help line to select it, and drag it to a new location. Press the Shift key if you want to lock the dragging at the original line angle.

In the second method, double click on one of the vertexes of a help line. The Help Line dialog box occurs displaying the current coordinates. Change a suitable number of values, and press OK

Joining Help Lines.

Two help lines can be joined by calculating the intersection point between the lines, if such a point can be found.

Click on the Help line tool. Select at least one point from each of the help lines to be joined. Select the Join Help Line command in the Tool menu. The lines are joined.

Locus Tool. 

The Locus tool is used for creating 3D locus points.

A locus point is a tree dimensional point or location used as a snap point for drawing 3D polygons. A locus point can only be modified when the Locus tool is active.

Another example of use is to draw two locus points with exact coordinates to provide an accurate positioning when using the Cut tool.

A third example of use is to place a locus point at a vertex in a mesh, which you need to take apart. This can easily be done by using the Selection tool, and setting the selection mode to Vertex mode. Drag all vertexes away from the locus point, one by one. The locus will remain at the original location because it can only be moved with the Locus tool. When the problem is fixed, simply drag the vertexes back to the location of the locus.

The Locus tool can only be created in the Front, Top, and Side views, but can be seen and snapped to in the Perspective view. Locus points can be printed on a printer if they are visible on the screen. To hide locus points, select Hide Loci in the Tool menu. Locus points are not included when exporting models to external formats.

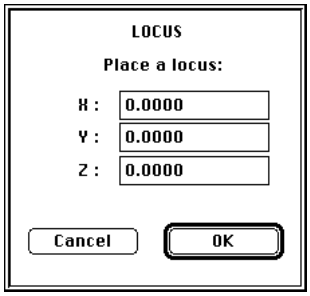
Using the Locus Tool.

To use the Locus tool, click on the Locus tool on the Tool palette. There are several ways to draw a locus:

In the first method, create a locus by simply clicking on the screen at a suitable location. A locus point is placed where you clicked. The location of the locus, in the direction you don't see in the current view is determined by the value in the Data Display bar. In the Front view, the location is determined by the Z value. In the Top view, the location is determined by the Y value. In the Side view, the location is determined by the X value. You can pre-set the depth location for the locus you intend to create, by entering a new coordinate value suitable for the current view.

An exception to the rules of depth location is when you click on a vertex while the cursor indicates a snap (2D or 3D). In such case, the locus will be placed at the location of the snap in 3D. If there are several possible snap locations in the depth direction, the locus will be placed at the vertex located closest to the coordinate displayed in the Data Display bar.

A second method to place a locus is to double click on the Locus tool. The Locus dialog box occurs, prompting you to enter coordinates for the Locus.



A third method to place a locus is to select the New Locus command in the Tool menu. The Locus dialog box occurs, prompting you to enter coordinates for the locus.

Selecting Locus points.

Several locus points may be selected by drawing a selection box, with the Locus tool. All locus points enclosed by the selection box will be selected. Another method for selecting several locus points is to shift click on different locus points.

Selected locus points may be deleted by pressing the Delete key. All locus points in a drawing may be deleted by using the Clear all Loci command in the Tool menu.

Moving Locus points.

A locus point may be moved by double-clicking on it. The Locus dialog box appears prompting you to enter coordinates for the locus.

Polygon tools

The polygon tools are a group of tools used for drawing polygons in four different ways.

Polygon tool. 

The Polygon tool is used for creating rectangles, parallelograms, and radial extrusions.

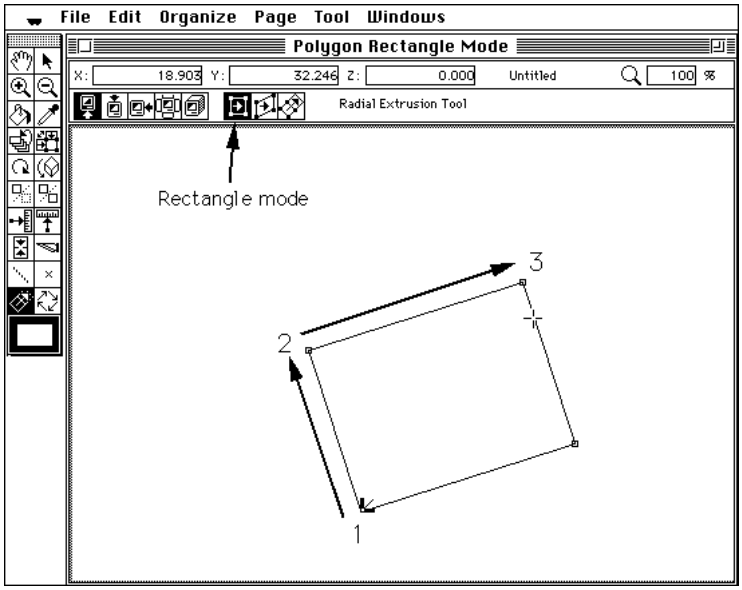
The Polygon tool has three drawing modes, each of which may be selected in the Mode bar.

Rectangle mode. 

The left hand mode button in the Mode bar is used for drawing rectangles.

Rectangles can be drawn tilted at any angle. Note that a rectangle oriented at 0°, 45°, 90°, and so on, may be drawn by pressing the Shift key before clicking a second time.

The Rectangle tool can be used in Front, Top and Side views but not in the Perspective and Unfold views.



Drawing in Rectangle Mode.

To draw a rectangle, select the Polygon tool, and then click on the Rectangle mode button in the mode bar. Move the cursor to the drawing area, and click at a

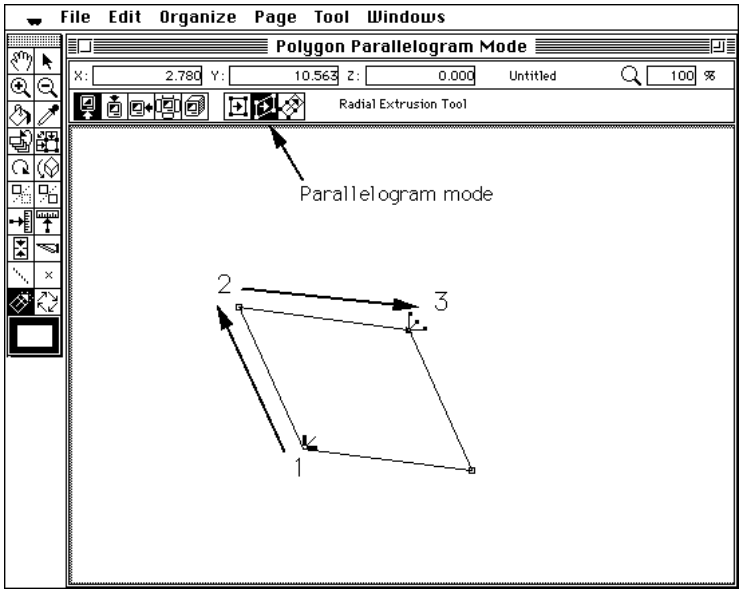
starting point (1). You don't have to press the mouse button while moving to the next step. In the next step we will establish the height and tilt of the rectangle. Move the cursor upwards on the screen, and click to establish a second point (2). In the next step we will set the width of the rectangle. This is done by moving the mouse, for example to the right. When a rectangular shape has been formed, click to complete the sequence (3).

Parallelogram mode. 

The center mode button in the Mode bar is used for parallelogram drawing.

Parallelograms may be drawn tilted at any angle. Note that a parallelogram base line oriented at 0°, 45°, 90°, and so on may be drawn by pressing the Shift key before clicking the second time.

The Parallelogram tool can be used in Front, Top and Side views but not in the Perspective and Unfold views.

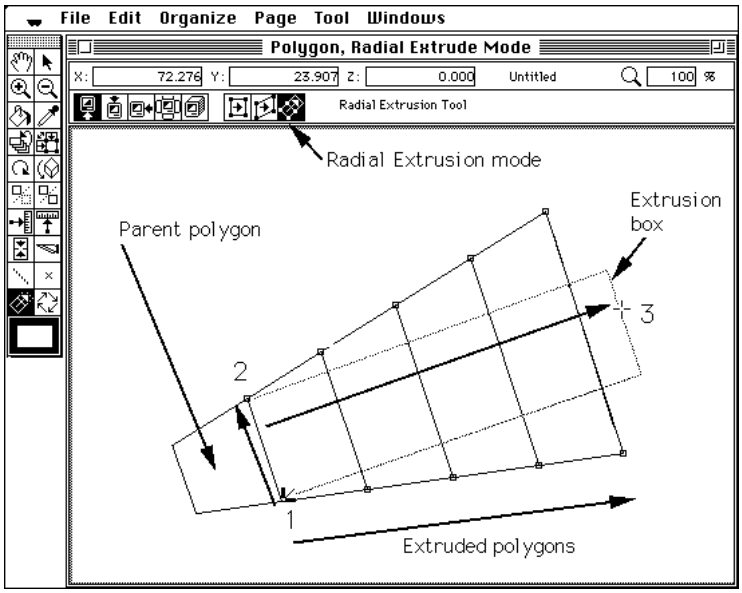


Drawing in Parallelogram mode.

To draw a parallelogram, select the Polygon tool, and then click on the Parallelogram mode button in the Mode bar. Move the cursor to the drawing area, and click at a starting point (1). You don't have to press the mouse button to move to the next step. In the next step we will establish the height and tilt of the parallelogram. Move the cursor upwards on the screen, and click to establish a second point (2). In the next step we will set the width and skewing of the parallelogram. This is done by moving the mouse, for example to the right. When a parallelogram shape has been formed, click to complete the sequence (3).

Radial Extrusion mode.

The right hand mode button in the Mode bar is used for extruding polygons sideways away from a selected polygon. One or several polygons may be extruded at a time. If the polygon you extrude from is not a rectangle, the extrusion lines will move away from or towards one another. This is true in all three directions. A unique feature with the Radial Extrusion tool is that it works in all views, including the Unfold view. The ability to extrude in the Unfold view is particularly useful for cardboard package design, because most cardboard boxes consist of several overlapping layers of material, creating an assembly mechanism. Said mode may also be used for placing assembly tabs at a location where they do not overlap the basic pattern.



Drawing in Radial Extrusion mode.

To use the Radial Extrusion tool, select a polygon to extrude from using the Selection tool on the Tool palette. Note that all vertexes of the polygon have to be selected. Select the Polygon tool on the Tool palette and click on the Radial Extrusion mode button in the Mode bar. Click on one of the selected vertexes (1). Click on a second vertex, which must not be located diagonally from the first vertex (2). Drag the cursor away from the parent polygon to a point located beyond the same distance as the width of the parent polygon. An extrusion box occurs, which is used for defining the number of new polygons to be created. Click to finish the sequence (3). If no new polygon occurs, it probably means that the extrusion box in the third sequence was too short. If two polygons occurs, it means that two parent polygons would fit in into the extrusion box. In the illustration above, four new polygons fitted into the extrusion box.

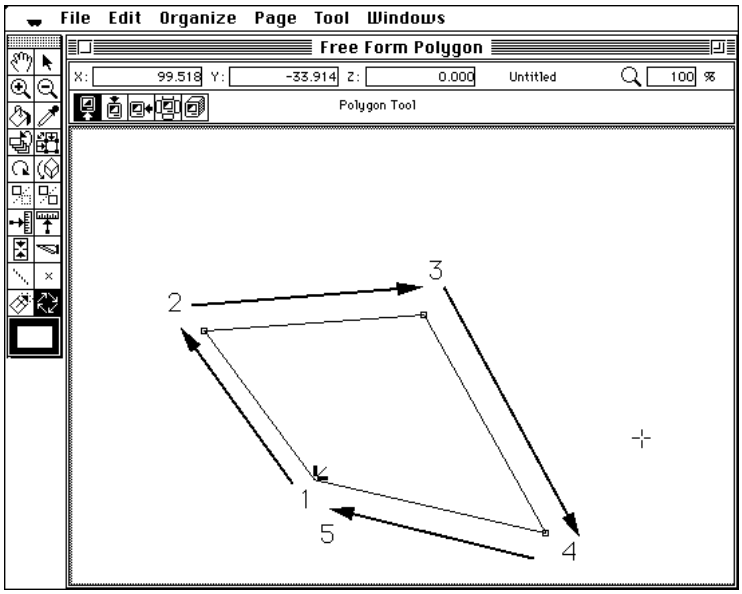
Free Form Polygon tool.

The Free Form tool is used for drawing free form polygons.

The Free Form tool works in Front, Top, Side and Perspective view, but not in the Unfold view.

Using the Free Form Polygon tool.

To use the Free Form Polygon tool, click on the Free Form Polygon tool on the Tool palette. Draw a polygon by clicking out four vertexes (1-4). Note that you do not have to press the mouse button while moving the cursor to the next vertex location. When you have clicked out the fourth vertex, the polygon automatically closes (5). If you double-click on the third vertex, or click on the starting vertex while being on the third vertex, a triangle will be created.



Line and Fill colors

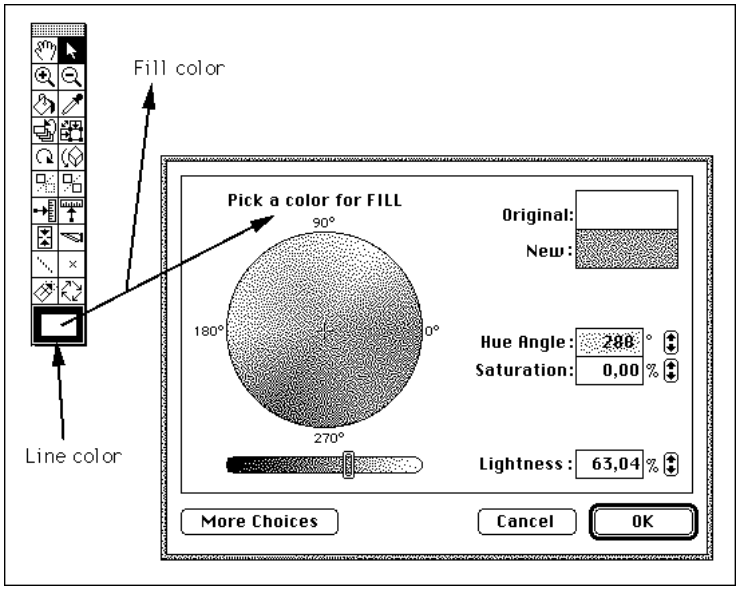
Line and Fill colors.

The Line and Fill Color tools are used for selecting line and fill colors for the polygons in the model. Colors can be selected either by using the Color Selection tool at the bottom of the Tool palette, or by using the Pick Up color tool. If you want to set the fill color to transparent, this can be done by using the Make Transparent command in the Tool menu. Transparent polygons can be filled by using the color tools on the Tool palette. New polygons use the current color settings.

Please refer to the Pick Up Colors and Put Down Colors tools for further information on how to change colors on polygons.

Using the Fill Color tool.

Click at the center part of the Color tool. A standard Macintosh Color selection dialog box occurs. Select a color.



Using the Line color tool.

Click on the border of the color tool. A standard Macintosh Color selection dialog box occurs. Select a color.

Command menus

Menu bar

There are six menus in Touch-3D™. They provide instructions for your Macintosh to carry out a desired action such as creating a new drawing, saving drawings, etc.

When you click on an item on the Menu bar, a menu of commands opens. Notice that some command names are black and others are grey. As you work, the appropriate commands for the current mode or tool are made available in the menus. The available commands are displayed in black; unavailable commands are dimmed. As you move the mouse pointer down the menu, command names will darken, or highlight. Commands are chosen by releasing the mouse button over a highlighted name in the menu.

Some menu commands have a small triangle on the right side of the menu, indicating that it has a sub menu for further choices. To select from these additional options, drag the mouse pointer to the desired command, and then drag towards the triangle. The additional options will then appear in the sub menu that drops from the first.

Other command names in the menus are followed by three periods. When selected, these commands will open a dialog box for you to enter data for the command.

File Menu

New (Command-N)

New is a command used for creating a new empty drawing.

Open... (Command-O)

Open is a general command for opening files. Files created in Touch-3D™ format, or other compatible files may be opened. When importing files that contain 2D and 3D information, it's recommended to use the Import-> options instead. Otherwise, Touch-3D™ will have to decide if the file is a 3D file or a 2D file. If problems occur when importing external file formats, please refer to the Import-> section of this chapter.

When the Open command (Command-O) is chosen in the File Menu, a directory dialog box is opened. The scrolling list in the dialog box contains all the available files on the current disk. If you click on the folder name, a menu will show the path from the current folder to the hard disk.

Close (Command-W)

The Close command (Command-W) in the File menu will close the active drawing. This has the same effect as clicking on the close box in the upper-left corner of the title bar.

If changes have been made that haven't been saved, you will be asked if you want to save them before the drawing closes.

Save (Command-S)

The Save command is used for saving the active drawing. If the active drawing hasn't been saved before, a standard Save dialog box will occur.

Save As...

The Save As... command is used for saving the active drawing, under a new name. When selected, a standard Save dialog box occurs.

Save Object As...

The Save Object As... command is used for saving selected polygons as a separate drawing. This command is useful for creating library components, which may be imported into other drawings.

Save Unfold As...

The Save Unfold As... command is used for saving selected unfolded polygons as a separate drawing. Unfolded models can be imported back to the 3D level for layout editing and optimizing the use of material.

Save Settings

The Save Settings command is used for establishing default settings, used when opening a new drawing. The default settings include drawing units, snap grid, paper settings, and filters.

Revert To Saved

The Revert To Saved command is used when you need to return to the last saved version of the active drawing.

Place...

The Place command is used for importing external models or components into the currently active drawing. Touch-3D™ will automatically extract any usable information from the external drawing. Touch-3D™-, DXF, MiniCad text-, EPSF, Spreadsheet, and QuickDraw 3D files may be used. When importing files containing 2D and 3D information, it's recommended to use the Place Import-> options instead.

Place Import ->.

The Place Import command is used for importing external models or components into the currently active drawing, using a specified file format. Touch-3D™ will automatically extract any usable information from the external drawing.

Export ->.

The Export commands are being used for converting the currently active drawing into external formats. Touch-3D™ can export as 2D & 3D DXF, 2D & 3D MiniCad text-, EPSF-, spreadsheet data, VRML, and 3DMF. When exporting a model in 2D, it is exported as seen on the screen. When exporting a 3D model, it is exported as a folded 3D model, except when exporting it in the Unfold view. It's then exported as a 3D unfolded model.

In some cases dialog boxes occurs when using the Export commands. In the MiniCad text case, the options are to export Touch-3D™ layers as MiniCad groups, layers or classes. In the DXF and 3DMF case the options are whether to use Y or Z oriented upwards in the coordinate system (check which applies to the receiving program), if you want to mirror the Touch-3D Z axis, whether to use four or three sided polygons, and, in the 3DMF case whether to export as 3D polygons or as a 3DMF Mesh. Use four sided polygons if possible because it reduces the file size. Use triangles for programs unable to handle twisted shapes, or when the receiving program renders models incorrectly.

Import ->.

The Import command is used for importing external models or components into a new drawing. In most cases the Open command will do the same thing, and automatically determine which file format is being used. When importing files containing 2D and 3D information, it is recommended to use the Import... options.

DXF, MiniCad text-, EPSF-, spreadsheet data, and 3DMF files may be used. Before importing a file, you should first determine its file type. Make sure that the program you want to export from supports at least one of the formats supported by Touch-3D™.

Touch-3D™ uses a very specialized drawing method. It's therefore necessary to make sure that the imported model is compatible with Touch-3D™. All elements in a model have to consist of either three or four-sided closed polygons. Other types of objects will automatically be ignored by the program. If you run into problems with the import function you will find a detailed description of what to do under the "Communication" chapter in this manual.

Touch-3D™ can import both 2D and 3D files. It may sound strange to import a flat drawing into a 3D program. Why unfold something that is already flat? The reason is that when a 2D drawing is imported into Touch-3D™, it's automatically converted into 3D. You are then able to drag any vertex in any direction to create a real 3D object.

When the Import command is chosen in the File Menu, a directory dialog box is opened. The scrolling list in the dialog box contains all the available files on the current disk. If you click on the folder name, a menu will show the path from the current folder to the hard disk.

Some notes on the import formats:

MiniCad 2D & 3D Text

This format is the text format of MiniCad. The Text format is not to be confused with Text created with a word processing program. When you work with MiniCad text files you have to set the units to one of the available decimal units. Do not use fractional units.

DXF 3D & 2D

The Data Exchange Format (DXF) is the de facto standard for CAD programs on both the Macintosh and DOS platforms. It was originally created as a data exchange format for AutoCad™. The text based version of this format can easily be exchanged between the Macintosh and DOS platforms.

EPSF

The Encapsulated Postscript Format (EPSF, EPSF, EPS) is a high resolution format that was created by Adobe. Both Printer and Adobe Illustrator EPSF files are supported. This format can easily be exchanged between the Macintosh and DOS platforms.

3DMF

The 3DMF format is Apple's 3D Meta File standard file format for conversion between 3D programs. Touch-3D™ supports polygon and mesh based models of this format. Compatible data types are polygons, general polygons, triangles, and 3DMF mesh models. 3DMF groups are automatically placed in separate layers.

Spreadsheet data

Spreadsheet data can be imported and exported. The format used must be TEXT based and use the TAB character as delimiter. Most spreadsheet and data base programs can handle these formats. It is also easy to create such a file using a word processor. Below is an example of how it should be formatted:
X (TAB) Y (TAB) Z (TAB) (New line), etc. In this case Touch-3D™ generates 3D locus points which can be used for skinning a model. If you use three or four groups of X,Y,Z coordinates per line Touch-3D™ will automatically generate a polygon based on these coordinates.

VRML

The VRML format is a cross platform format mainly used for displaying interactive 3D models on Internet. VRML models can for example be displayed using The Netscape Navigator plug in Live3D which can be downloaded from the Netscape web page.

Page Setup...

The Page Setup command in the File menu specifies the page size and orientation of the current drawing when it is printed. It is important to remember that the information displayed in the Page Setup dialog box varies with the printer currently selected in the Chooser and with the Macintosh used. It's also possible to print out a model over several sheets. This is controlled by the menu command Printing Size... in the Page menu. The model may be re-centered on the sheet by using the menu command Move Paper in the Page menu. For further information on this subject, please refer to these commands later on in this manual. The drawing can also be scaled up and down to fit into the drawing by using the Printing Scale command in the Page menu. This command adds a scaling factor to the model while displaying it on the screen and when printing, but it does not change the size of the model or it's coordinates.

Print... (Command-P)

The Print command will print or plot a copy of the active drawing. Touch-3D™ will print or plot to any Chooser level printing device. All visible objects in the drawing window, will be printed. To use a pen plotter or any other HPGL compatible device you need a plotter driver program. Touch-3D™ works with any of the available drivers on the market.

Quit (Command-Q)

The Quit command (Command-Q) will close Touch-3D™. If any of the open drawings has been modified, you will be asked if you want to save them. If there are several modified drawings, you will be asked about each one individually.

Edit Menu

Undo (command-Z)
The Undo command is used for undoing the last performed action. Only actions affecting vertexes can be undone.

Cut (command-X)
The Cut command is used for cutting out polygons from Touch-3D™. These objects can be pasted back into any Touch-3D™ drawing as 3D objects.

Copy (command-C)
The Copy command is used for copying polygons from Touch-3D™. These objects can be pasted back into any Touch-3D™ drawing as 3D objects, or into other applications as PICT files.

Paste (command-V)
The Paste command is used for pasting polygons into Touch-3D™ drawings. They are placed at their original coordinates. When pasting objects into other programs, the PICT file format is used. Note that you can not paste PICT based objects from other programs. PICT files can only be imported into Touch-3D™ using the Load Background Picture command in the Tool menu.

Clear
The Clear command is used for deleting selected polygons from the currently active drawing.

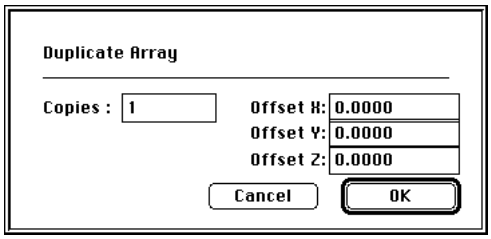
Select All (command-A)
The Select All command will select all available polygons in the current drawing. Note that if the Layer visibility is set to Show/Snap/Modify Others, all polygons are selected, if they are located in a visible layer. If not, all objects in the currently active layer are selected.

Duplicate (command-D)
The Duplicate command copies any selected polygons and places them slightly to the right, and below the original(s), as seen in the currently active view.

Duplicate Array...

The Duplicate Array command is used for creating multiple copies of the currently selected polygons, at a given offset from one another.

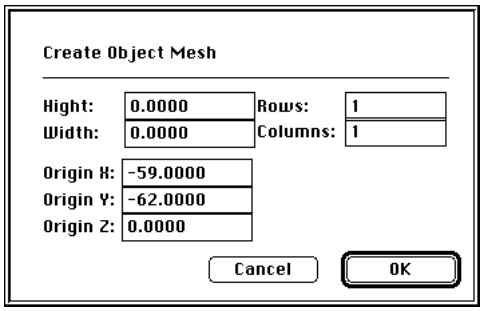
Select the number of polygons you want to copy. Note that all corners of a polygon have to be selected to be considered activated in this command. Select the Duplicate Array command. A dialog box occurs where you are prompted to enter the required number of copies and the vertical, horizontal and depth offsets. Click OK, and the new copies will be created.



Create Object Mesh...

The Create Object Mesh command is used for creating a grid of rectangles, often used as a starting point for free form modelling.

Select the Create Object Mesh command. A dialog box occurs where you are prompted to enter the size of the basic rectangle, and the required number of copies, vertically and horizontally. The origin point is always placed at the bottom left corner of the mesh, regardless of creation view.

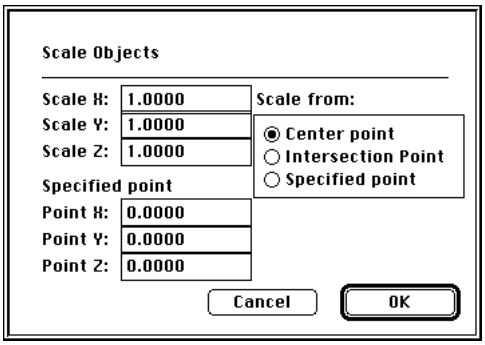


Scale Objects...

The Scale Objects command is used for rescaling selected vertexes or polygons by adding a scale factor. The factor 1 indicates that no scaling is to take place. A factor of 0.1 indicates scaling down to one tenth of the original size, whereas 10 indicates scaling up to ten times of the original size. Please note that you also have the option to resize by absolute values using the Reshape... menu command.

To use the Scale Objects command, select a number of polygons or vertexes. Select the Scale Objects command. Enter a scale factor for X, Y, and Z, select the

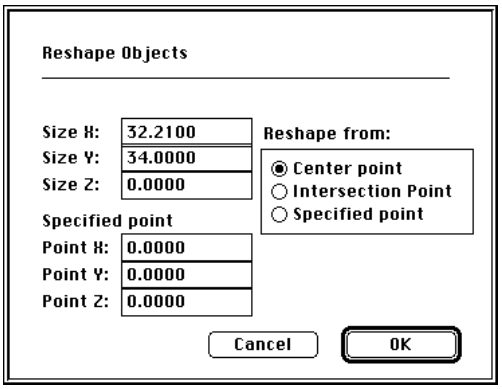
point from which the scaling is to take place. There are three options; The first is to use the center point which is the physical center point of the selected vertexes. The second option is to scale from the Intersection Point. The third option is to type in the coordinates for a scaling point in the Point X, Point Y, Point Z edit fields. Click OK to perform the command.



Reshape... (command-R)

The Reshape command is used for resizing selected vertexes or polygons by specifying the size of an "enclosure cube", defined by the minimum and maximum coordinates of the selected vertexes in their respective direction. Please note that you also have the option to resize by using a scaling factor, found in the Scale Objects... menu command.

To use the Reshape command, select a number of vertexes. Select the Reshape... command. A dialog box occurs, stating the present size of the "enclosure box". Enter new X, Y, and Z dimensions, select the point from which the scaling should take place. There are three options; The first is to use the center point, which is the physical center point of the selected vertexes. The second option is to use the Intersection Point. The third option is to type in coordinates for a scaling point, which is done in the Point X, Point Y, Point Z edit fields. Click OK to perform the command.

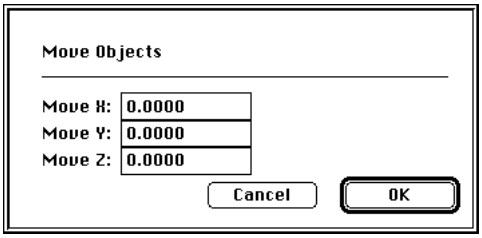


Move... (command M)

The Move command is used for moving selected vertexes or polygons a given 3D distance.

To use this command, select a number of vertexes. Select the Move... command. A dialog box occurs prompting you to enter offset distances in the X, Y, and Z directions. Press OK. The vertexes will now move the stated distance, in their respective directions.

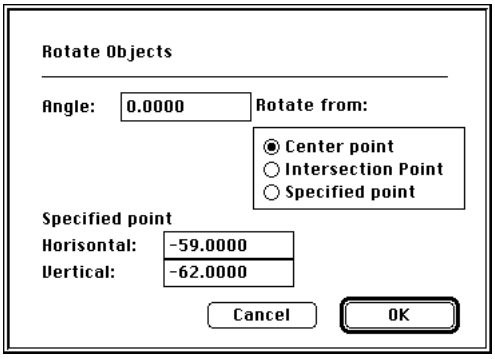
Note a similar command is available in the Data Display bar, by typing the letter M or m before the specified moving distance.



Rotate... (Command-T)

The Rotate command is used for rotating any number of selected vertexes or polygons, a specified number of degrees. The rotation is strictly two dimensional, and works as seen in the present view.

Select a number of vertexes. Select the Rotate... menu command. A Dialog box occurs prompting you to enter a rotation angle. Vertexes may be rotated either around the center point, around the Intersection point, or around a point stated by the two coordinates in the bottom left edit fields. Click OK to perform the rotation.



Rotate Left 90° (Command-L)

The Rotate Left 90° command is used for rotating any number of selected vertexes or polygons 90 degrees counter clockwise, around the center point of the objects, as seen on the screen.

Rotate Right 90°

The Rotate Right 90° command is used for rotating any number of selected vertexes or polygons 90 degrees clockwise, around the center point of the objects, as seen on the screen.

Flip Horizontal

The Flip Horizontal command is used for mirror rotating any number of selected vertexes or polygons horisontally, around the center point of the objects, as seen on the screen.

Flip Vertical

The Flip Vertical command is used for mirror rotating any number of selected vertexes or polygons vertically, around the center point of the objects, as seen on the screen.

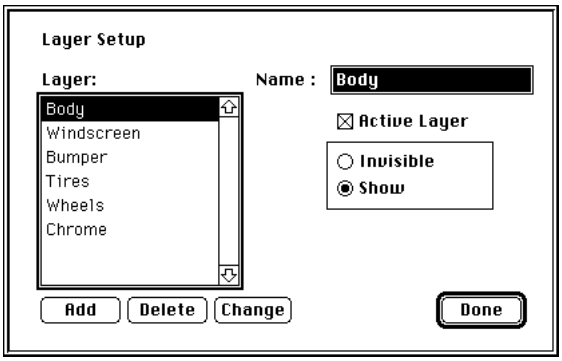
Mirror in Unfold

The Mirror in Unfold command mirrors the selected object group in the Unfold view. It is primarily used for correcting objects that come out mirrored when unfolded. This usually occurs when symmetrical objects are being created. In some cases you may up having for example two left sides and no right side.

Layer Configuration...

The Layers Configuration command is used to add, delete, rename, activate and hide drawing layers. Layers are used to divide the model into sub-components. This is useful in several ways. Parts of the model may be hidden, while working with other parts. Layers also divide the models into components that are unfolded separately which speeds up calculations considerably. Layers are also used for creating a good structure, when exporting and importing to and from other programs, since most CAD and modelling programs use layers to divide models into sub-components.

Select the Layers Configuration command. A dialog box occurs.



To the left there is a list of the available layers.

A layer is added by typing in a new name in the edit field to the right, and then by pressing the Add button.

A layer is renamed by clicking on any of the available layers in the list to the left. The name is highlighted, and is also displayed in the edit field to the right. Type in a new name, and click on the Change button to change the name.

A layer is deleted by clicking on any of the available layers in the list to the left. The name is highlighted, and is also displayed in the edit field to the right. Make sure that you really want to delete the layer, and it's contents. Press the Delete button to delete the layer. Note that you can't undo this command.

Layers can either be visible or invisible. This is done by selecting a layer in the list to the left, and then pressing either the visible, or invisible radio button to the right. Layers defined as invisible are not lost, they are just invisible, and can be made visible again. This function can reduce screen clutter, and increase the speed of screen updates. All layers may be defined as invisible, but the active layer will always remain visible, regardless of setting.

The Layer configuration dialog also has a check box for selecting the active layer. The active layer is the layer where new objects will be placed, when they are drawn. To change the active layer scroll to the right layer, select it, and click on the Active Layer check box to select it.

Active Layer ->

The Active Layer command is used for changing active layer. To change active layer drag the mouse to the Active Layer command, and then to the new active layer name.

Set Layer ->

The Set Layer command is used for moving polygons from one layer to another. Select a number of polygons, select the Set Layer command, and then go to the new layer name for the polygons. The polygons will now be moved to the chosen layer.

Layer Visibility

The Layer Visibility command is used for setting the program in a different display mode. There are four modes; In the first mode (Active Only), only objects located in the currently active layer are displayed on the screen. In the second mode (Show Others), all objects located in layers defined as visible are shown, but only objects in the active may be modified. In the third mode (Show / Snap Others), objects located in layers defined as visible are shown, and may be snapped to, but only objects in the active may be modified. The fourth mode (Show / Snap / Modify Others), allows full editing in any layer, defined as visible.

Perspective

The perspective command is used for distorting the model to make it look more natural. When modelling, it's recommended that the perspective should be turned off. To turn the Perspective mode on and off, select the Perspective command.

Hidden Lines

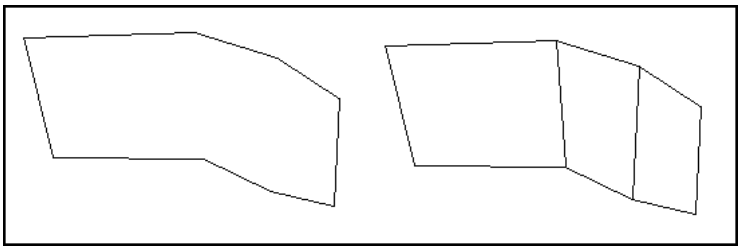
The Hidden Lines command is used for displaying the model with hidden lines removed. This function is only intended as a quick orientation aid, not for final renderings. Complicated models may frequently be rendered incorrectly. Still, this is a good aid for understanding the shape of a model.

For more adequate renderings, we recommend a dedicated rendering program.

The Hidden Lines command also works in the Unfold view where it uses the fill color setting used in the 3D model. If you combine the Hidden Lines on setting with the OutLine on setting you get a color model with only outlines visible which is useful for printing ready to use color kits.

Outline

The Outline command is used for eliminating unused intermediate line in an unfolded model. This makes the pattern less cluttered and allows you to generate a ready to use cut out pattern. Such a pattern can also be exported in MiniCad text, DXF and Illustrator format for use in a cutting or milling machine. As always, models are exported as seen on the screen.



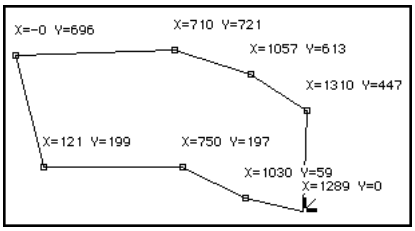
The precision of the outline calculation, that is how close vertexes have to be to be considered being located at the same point, is controlled by the Filter command in the Page menu. If the outline displays small gaps, for example at the end of a slot, try increasing the precision in the Filter dialog.

Show Unfold Coordinates.

The Show Unfold Coordinates command allows you to display the coordinates of any number of vertexes in the Unfold view. This can be useful when manually transforming a scale drawing into a full scale production template of a large object such as a boat hull, a sculpture, tents, hot air balloons, etc.

To display the coordinates you simply select the Show Unfold Coordinates option, and select any number of vertexes in the Unfold view. The coordinates are displayed in nine points font size on the screen and at six points when printed or exported as DXF, MiniCad text or as an Illustrator file. The coordinates are therefore always readable on the screen, and as small as possible to save space when printed or exported. The number of decimals displayed is controlled by the Set Grid command in the Page menu.

To export a multi layer model, set the Layer visibility option in the Organize menu to "Show, Snap and Modify Others" and select all vertexes. This allows you to export the content of all visible layers together with the coordinates placed in their respective groups/layers/classes.



Fit Model To Window

The Fit Model To Window command is used for changing the zoom factor and centring the model to fit within the current drawing surface.

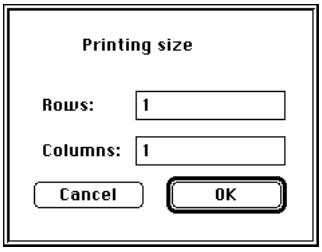
Fit Paper To Window (Command-4)

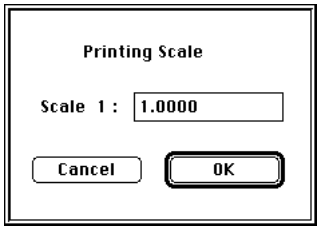
Fit to Window (Command-4) in the Page menu sets the scale to 100% and places the drawing surface at the center of the screen. This is how the model is displayed when you open a document.

Printing / Paper ->

-> Printing Size...

The Printing Size command provides a method for printing over several sheets of paper. If the paper orientation is set to tall in the Page Setup dialog box, and the Paper size is set to A4 / A, and you set Rows to one, and columns to two in the Printing Size dialog, you will get an A3/B size paper distributed over two sheets of paper. Note that the grey frame on the drawing surface indicates the printable part of the paper, which is smaller than the total size of the paper. If you set the printing size to for example two sheets of paper, the drawing surface will contain two grey frames, indicating that you work on two sheets of paper. When a drawing has been printed over several sheets of paper, the sheets may be joined into a single sheet of paper by cutting of the papers where the printing stops, and then taping or gluing them together.



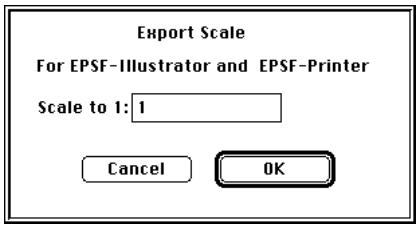


->Printing Scale...

The Printing Scale command is used for fitting the model into the paper. As with most CAD programs, Touch-3D™ uses full scale measurements when modelling. If the a polygon is 3000 x 5000 mm, you simply type in these values. A Printing Scale of 1:20 makes it look 150 x 250 mm, and may be printed out on a single letter sized sheet of paper.

->Export Scale...

The Export Scale command is only used by the EPSF Export commands. The reason is that some illustration programs lack a drawing scale function. Touch-3D™ always uses full scale. If you for example want to fit a car into a normal drawing sized sheet of paper you have to set the scale to something about 1:20. (E.g. 4 meters = 20 centimeters)



->Move Paper.

The Mover Paper command is used for moving the grey printing frame to a location where it encloses the model. This is of course required to be able to print out the model. Since Touch-3D™ is not a 2D program, the Move Paper function has to be set individually for the Front, Top, Sid, Unfold, and Perspective views.

->Center Paper

The Center Paper command is used for automatically performing the Move Paper command.

-> Fit To Model

The Fit To Model Command sets the Printing Scale suitable level for printing, when you need to make the model as big as possible, while still fitting into the current paper size setting.

->Show Outline Only

The Show Outline Only command turns the visibility of the page individual breaks on and of. This only applies when the Printing Size command is set to more than one paper.

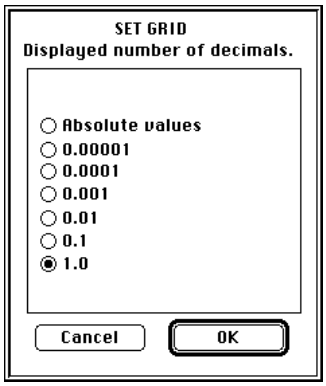
->Hide Paper

The Hide Paper command allows the grey paper frame to be turned on and off.

Set Grid...

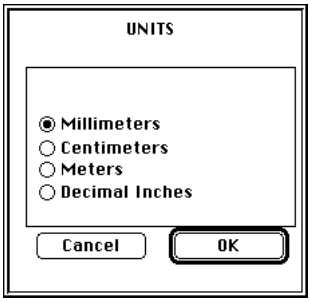
The Set Grid function sets the snap grid. If you for example set the snap grid to 1,0 Touch-3D™ will place vertex points at coordinates rounded to the closest non decimal value, when placed without snapping.

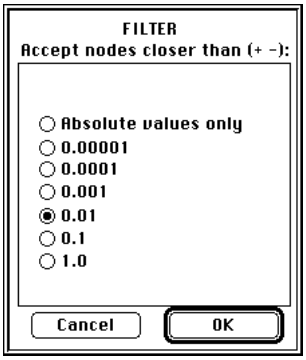
The Set Grid command also affects the length of the steps used by the Arrow keys. When you have selected a vertex / Intersection point, you are able to move it in any direction by pressing the arrow keys, or the plus key (+) for moving towards you, and minus key (-) for moving away from you.



Units...

The Units dialog controls the drawing units used (e.g., mm, meters, decimal ft. and inches) in your drawing.





Filters...

Decimal values are usually rounded to some extent. When Touch-3D™ reads files, these roundings may cause errors when building models. In order to be able to connect vertexes to one another Touch-3D™ sometimes needs to round off the values. The Filters command sets the sensitivity for this. In theory 1.00001 is not the same as 1.00000, but close. If you set the Filter to for example 0.0001 Touch-3D™

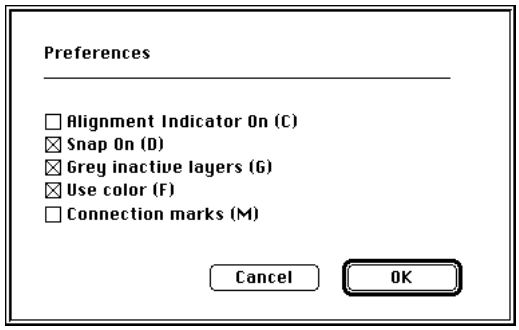
will interpret these values as equal. Normally it's good practice to set the Filter to a slightly lower number of decimals than you have in the files you read.

Rebuild Model

The Rebuild command rebuilds the model in the same way as Touch-3D™ builds models when you import them into the program. It may be necessary to use this command in some cases when you edit models and join elements. If you don't, the elements of a model may be treated as separate objects when moved although they are connected. Use this command when it happens.

Preferences...

The Preferences... command is used to set various program settings. All these settings can be toggled on and of by pressing a key (indicated within parenthesis)



-> Alignment Indicator On (C)

This function is used for creating imaginary help lines. To use it, hold the cursor over a certain vertex without pressing the mouse button. If you drag the cursor to the right-, left-, up-, or downwards, while remaining roughly aligned with the original vertex, the cursor will indicate an alignment by displaying a dot at the tip of the aligning direction of the cursor. The selected vertex is being indicated by a circle.

The Alignment Indicator function can store up to two vertex locations at a time, which for example can be used for calculating an intersection between two vertexes. In this case the cursor displays dots on both the vertical and horizontal axes.

Please refer to the Selection tool chapter of this manual for a more detailed description on how to use the Alignment Indicator.

-> Snap On (D).

The Snap On function toggles the automatic snap function on and off. If the function is off, the cursor won't indicate snapping to a vertex. This may be useful if there are many vertexes located very close to one another, and you still want to place a vertex without snapping to one of the others.

-> Grey Inactive Layers(G)

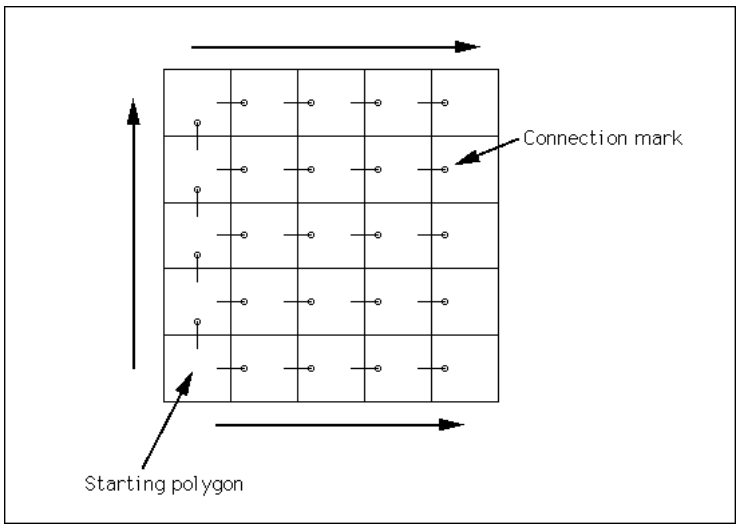
When the Grey Inactive Layers function is selected, inactive layers are displayed with more pale colors (greyish). This may be helpful because it suggests which objects that belong to the active layer.

-> Use Color(F)

The Use Color function allows toggling between displaying the model in color or not.

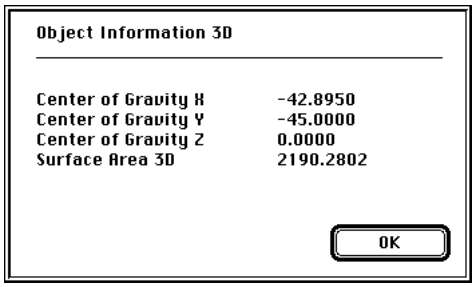
-> Connection Marks(M)

The Connection Marks function displays the order in which the polygons are connected mutually to one another at any given time.



Object Info... (Command-I)

The Object Info command is used for calculating the 3D surface area and center of gravity of any selected polygons. Note that all vertexes of a polygon have to be selected to be included in the calculation.

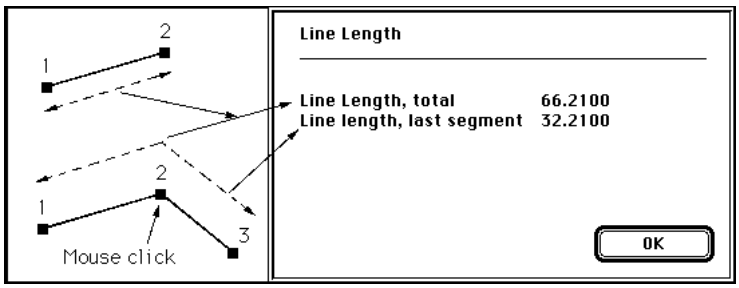


Make Transparent

The Make Transparent command is used for making selected polygons appear transparent, when displayed with hidden lines. To make transparent objects look solid again, select the polygon, and use the Put Down Color tool to change the colors.

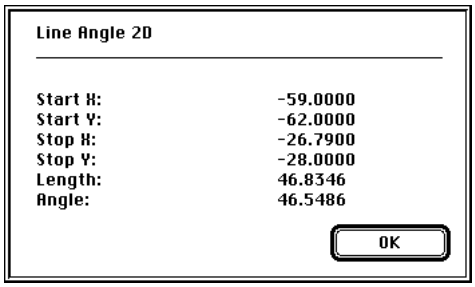
Line Length

The Line Length command is used for calculating the length of a line or of a polygon. Select the tool. Click out a line or a polygon on the screen. Double click to terminate the function. A dialog box showing the accumulated length, and the length of the last section of the line or polygon appears. The line or polygon does not have to follow the lines of the model, and may be drawn in any order.



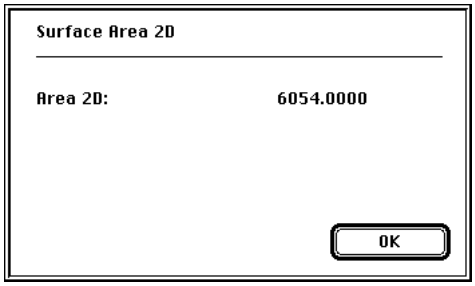
Line Angle

The Line Angle command is used for calculating the properties of a line, previously drawn on the screen. The result is displayed in a dialog box, appearing after the line has been drawn.



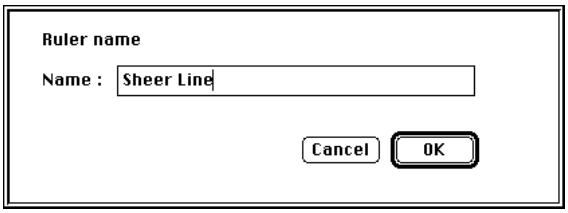
Calculate 2D Area

The Calculate 2D Area tool is used for calculating the 2D surface area of a polygon drawn on the screen. Click out a polygon on the screen. The function is terminated by either double clicking, or clicking at the starting point. In the first case, Touch-3D™ automatically closes the polygon, by drawing an imaginary line to the starting point. A dialog box appears showing the 2D surface area of the polygon. An example of use of this function is to calculate the front area of a car body, which for example is important for calculating the drag of a car body.



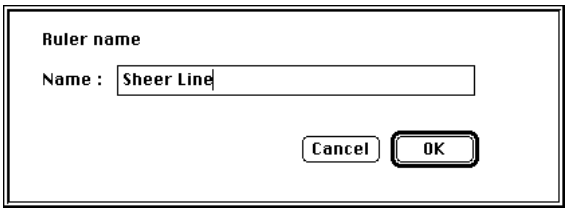
New Ruler

The New Ruler command is used for creating a new Alignment Ruler. Before using this command, make sure that you are in the correct view (Front, Top, or Side), and that you have selected either the Horizontal or Vertical Alignment Ruler Tool on the Tool palette. When done, select the New Ruler command in the Tool menu. A dialog box appears, prompting you to type in a ruler name. When the OK button is being pressed, the new ruler name will appear in the Select Ruler pop up menu.



Rename Ruler...

The Rename Ruler command is used for renaming previously named or unnamed rulers. Select the ruler to be renamed by using the Select Ruler ->(ruler name) command. Select the Rename Ruler command. A dialog box showing the old rulee name appears. Type in the new name, and click OK.



Select Ruler -> (ruler name)

The Select Ruler Command is used for changing active ruler.

Hide Rulers

The Hide Rulers command is used for hiding the rulers. When selected, the rulers remain in the drawing but are invisible. The rulers can be made visible again by reselecting the command.

Clear All Rulers

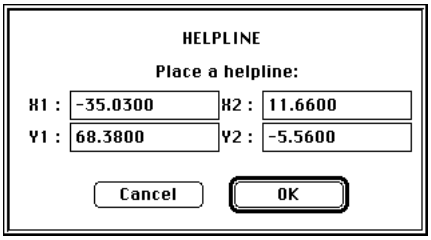
The Clear All Rulers command is used for deleting all rulers from the active drawing.

Lock To Rulers (Command-B)

The Lock To Rulers command is used to lock the selected vertexes to the active ruler. When the command is active, the selected vertexes will follow any movement performed with the ruler. Note that the movement does not push non-aligning vertexes to align the ruler, they follow the ruler in parallel. This is an important tool for deforming a model into a smooth and rounded organic shape. The Alignment Ruler chapter of this manual provides more information on how to use this command.

New Help line

The New Help line command is used for creating a new 2D help line on the screen, by typing in coordinate values. The command can also be selected by double clicking on the Help Line Tool on the Tool palette.



Hide Help Lines

The Hide Help Lines command is used for hiding the help lines. When selected, the help lines will remain in the drawing, but invisible. The help lines can be made visible again by reselecting the command.

Clear All Help Lines

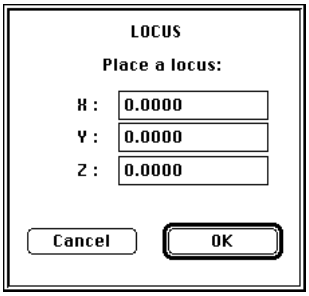
The Clear All Help Lines command is used for deleting all help lines in the active drawing.

Join Help Lines (Command-J)

The Join Help Lines command is used for calculating where two help lines intersect (if they do), and to stretch them to the point where they meet. This is a useful design tool when seeking "hidden" alignment points.

New Locus

The New Locus Point command is used for creating new locus point by typing in the coordinates. When the command is selected, a dialog box appears where the coordinate values are being typed in. Press OK, and the Locus will appear on the screen. Note that it may be invisible at the present degree of zoom and pan.



Hide Loci

The Hide Loci command is used for hiding locus points. When selected, the loci will remain in the drawing, but invisible. The locus points can be made visible again by reselecting the command.

Clear All Loci

The Clear All Loci command is used for deleting all Loci in the active drawing.

Load Background Picture

The Load Background Picture command is used for loading a background picture into the drawing. The picture has to be in PICT format. The background picture is used as a design assistant during outlining of a new model. Many designers find it useful to be able to place a simple sketch behind the model. This sketch may be drawn in another program or scanned in from a paper sketch.

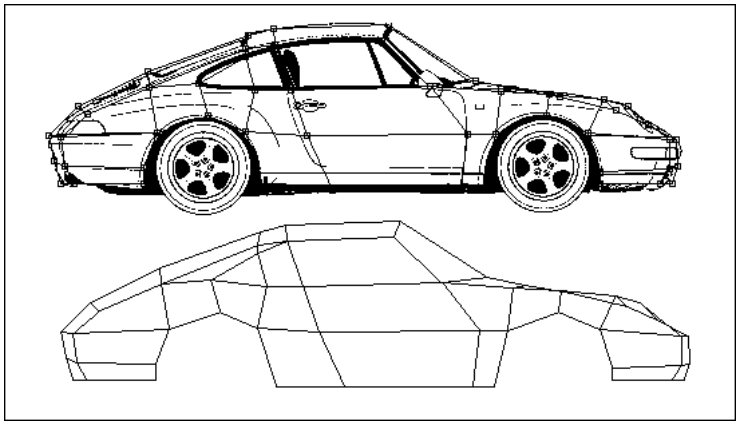
Our recommendation is to run the Fit to Window command before loading the background picture, and whenever you need to re-center the model over the picture. When you're done trace copying the picture, use the Reshape... command to resize the model to the correct size.

Hide Background Picture

The Hide Background Picture command is used for hiding the present background picture. This is useful because a big background picture may slow down the program substantially.

Unload Background Picture

The Unload Background Picture command is used to clear the currently active background picture from the memory.



Background picture.

Close All

The Close All command is used for closing all currently active drawings. If any of them has not yet been saved with a correct document name, you will be prompted to give it a correct name.

Save All

The Save All command is used for saving all currently active drawings. If any of them has not yet been saved with a correct document name, you will be prompted to give it a correct name.

Active files -> (names of active drawings)

The last item(s) in the Windows menu are used to select an active drawing other than the currently active. This is of course only possible if at least two drawings have been loaded into the memory.

Communication

Touch-3D™ can communicate with many external programs. It supports 2D & 3D DXF files (text based only), 2D & 3D MiniCad text files, EPSF (EPS) files (Printer and Illustrator files), and 3DMF (QuickDraw 3D, binary and text based). Since these files are all text based, except 3DMF binary, it's possible to import and export files to the DOS world by using the Apple File Conversion utility program, PC Exchange or any of the other conversion programs available on the market.

Model specification

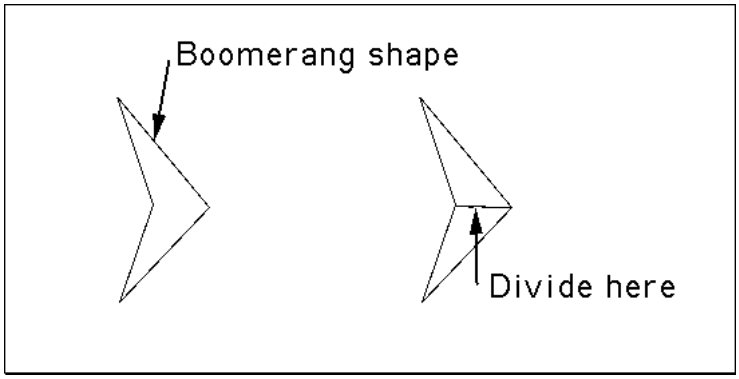
It is important to realize that the drawing structure of Touch-3D™ limits what you can import into it. Models have to consist of three or four-sided closed polygons. Touch-3D™ ignores all invalid elements. *We can not guarantee that your program will work with Touch-3D™ because your program has to be able to provide not only a correct type of file but also a correct type of drawing elements.*

Things to avoid:

• Avoid 3D polygons with five or more sides. These will be ignored by Touch-3D™. It may be confusing that some elements appear to have been lost. Divide these surfaces into smaller units.

• Avoid excessively twisted shapes (propeller shaped). This can cause substantial calculation errors. If possible, divide twisted surfaces into smaller units, because this will reduce the calculation error.

• Avoid boomerang shaped surfaces. It's better to divide the surface into smaller units. It may cause a calculation error.



• Touch-3D™ can handle and unfold several models at the time. However, it's good practice to avoid trying to unfold many elements that are not connected to one another. If you take for example 750 polygons were none of the elements are connected to one other a Quadra level will work for 10 - 20 minutes trying to combine the elements into something that makes sense.

• Avoid importing units not based on decimals. This is probably not a problem because most CAD programmes export DXF files as decimal inches, regardless of what you have set the programme to work in. They use this unit internally and convert it into whatever you set the programme to use.

• Avoid placing very large models in one layer. It's better to divide the model into several components, using layers, because Touch-3D™ does not try to combine surface elements located in different layers with one another. This means that the possible number of combinations is reduced dramatically.

Importing 3D files

A model can be imported in a number of ways. In some cases you may have to experiment with your program to get it to do what you want. In the following section we will specify each format individually.

Importing 3D DXF files

The first thing to do is to try to open the DXF file. If it works you can stop reading this section here.

If it doesn't work, try this:

The import file has to consist of a text based DXF file. In this case the elements should be defined by the DXF code "3DFACE" or "POLYLINE". To check whether your program really delivers the right code you can do as follows:

• Generate a DXF file.

• Open the file in a word processing program. (Any program that supports TEXT files will do)

• Locate the text string "3DFACE" or "POLYLINE" by using the Search function in the word processor.

• If you can find the string "3DFACE" or "POLYLINE" the file may be OK. If you still have a problem to import the file or if you loose some information you can check whether your information really consists of three- or four- sided polygons. Touch-3D™ ignores all "3DFACE":s or "POLYLINE":s that has five or more sides or less than three sides. The cure for this problem is to divide five- sided (or more) polygons to smaller units.

• If you can't find the string "3DFACE" or "POLYLINE" in the file, you have to check in the manual of your drawing program, whether you have a function that can

divide the model into polygons. Many programs, including AutoCad™, have a command for this. (MiniCad: "Convert to 3D polygons", AutoCad: "Explode")

• If you get more information than you wanted you may have some other information which can be read by Touch-3D™, for example in another layer. If the model is a part of a very large file, it might be a good idea to create a separate export file. Copy the model from the original file to the export file and then export the model from that file. In this way you avoid getting any unwanted information in your DXF file. Another advantage is that the file will be much smaller which means that it takes substantially less time to scan it for usable information.

Importing 3D MiniCad text files.

The first thing to do is to try to open a MiniCad text file. *When you work with MiniCad text files you have to set the units to one of the available decimal units. Do not use fractional units.* If it works you can stop reading this section here.

If it doesn't work, try this:

The import file has to consist of 3D polygons. This can be done in MiniCad as follows:

• Select the elements you want to convert to 3D polygons.

• Select the command "Convert to 3D polygons" in the "3D" menu (Delta menu on older versions).

• Select the command "Export as Text" in the File menu

• Import the model into Touch-3D™.

• If you get more information than you wanted you may have some other information which can be read by Touch-3D™, for example in another layer. If the model is a part of a very large file, it might be a good idea to create a separate export file. Copy the model from the original file to the export file and then export the model from that file. In this way you avoid unwanted information in your MiniCad text file. Another advantage is that the file will be much smaller which means that it takes substantially less time to scan if for usable information.

Importing 3DMF files.

Open a 3DMF file (QuickDraw 3D). Both text based and binary files are supported. If Touch-3D™ refuses to read the file it usually means that the file does not contain any type of information supported by Touch-3D™. Touch-3D™ supports 3DMF files defined as polygons, general polygons, triangles, and mesh based models.

Exporting 3D files

Touch-3D™ can export 3D files as DXF, MiniCad text, and 3DMF files. Before you start exporting the model you have to decide whether you want to export an unfolded or the 3D version of the model. The unfolded version can be exported as a 3D object with all coordinates in one direction set to zero.

If you want to export the 3D model you have to set the view to either the Front, Top or Side views. If you want to export the model as an unfolded 3D object you set the view to the Unfold view.

• Select the Export as.. (MiniCad 3D text), (3D DXF) or (3DMF) command in the File menu.

• Type in the file name in the standard Save dialog box.

• Import the file into your 3D program.

Importing 2D files

Touch-3D™ can import both 2D and 3D files. It may sound strange to import a flat drawing into a 3D program. Why unfold something which is already flat. The reason is that when a 2D drawing is imported into Touch-3D™, it is automatically converted into 3D. You are then able to drag any vertex in any direction and create a 3D object with a real depth.

A model can be imported in a number of ways. In some cases you may have to experiment with your program to get it to do what you want. In the following section we will specify each format individually.

Importing 2D DXF files

The first thing to do is to try to open the DXF file. If it works you can stop reading this section here.

If it doesn't work, try this:

The import file has to consist of a text based DXF file. In this case the elements should be defined by the DXF code “3DFACE” or "POLYLINE". To check whether your program really delivers the right code you can do as follows:

• Generate a DXF file.

• Open the file in a word processing program. (Any program that supports TEXT files will do)

• Locate the text string “3DFACE” or "POLYLINE" by using the Search function in the word processor.

• If you can find the string “3DFACE” or "POLYLINE", the file may be OK. If you still have a problem to import the file or if you loose some information you can check whether your model really consists of three- or four- sided polygons. Touch-3D™ ignores all “3DFACE”:s or "POLYLINE":s that has five or more sides or less than three sides. The cure for this problem is to divide five- sided (or more) polygons into smaller units.

• If you can’t find the string “3DFACE” or "POLYLINE" in the file, you have to check in the manual of your program whether you have a function that can divide the model into polygons. Many programs, including AutoCad™, can do this with a command in the program.

• If you get more information than you wanted you may have some other information which can be read by Touch-3D™, for example in another layer. If the

model is a part of a very large file, it can be a good idea to create a separate export drawing. You then copy the model from the original file into the export file and then export the model from that drawing. In this way you avoid getting any unwanted information into your DXF file. Another advantage is that the file will be much smaller which means that it takes substantially less time to scan it for usable information.

Importing 2D MiniCad text files

The first thing to do is to try to open a MiniCad text file. When you work with MiniCad text files you have to set the units to one of the available decimal units. Do not use fractional units. If it works you can stop reading this section here.

If it doesn't work, try this:

- The import file has to consist of closed 2D polygons.
- Select the command "Export as Text" in the File menu
- Import the model into Touch-3D™.
- If you get more information than you wanted you may have some other information which can be read by Touch-3D™, for example in another layer. If the model is a part of a very large file, it might be a good idea to create a separate export file. Copy the model from the original file to the export file and then export the model from that drawing. In this way you avoid getting any unwanted information into your MiniCad file. Another advantage is that the file is much smaller which means that it takes substantially less time to scan it for usable information.

Importing 2D EPSF files

Try to open an EPSF file. If it works you can stop reading this section here.

If it doesn't work, try this:

- You can't import EPSF files from Paint programs.
- The import file has to consist of closed 2D polygons.
- Export the file from the original file. Don't forget to specify that you want to export an EPSF file (EPS).
- Import the model into Touch-3D™.

Exporting 2D files

Touch-3D™ can export 2D files as DXF, MiniCad text and EPSF files. Before you start exporting the model you have to decide what you want to export. Touch-3D™ always exports 2D versions of the model as seen on the screen, including the outline and show coordinates options. If you for example want to export the unfolded version of the model you have to make sure that you are in the Unfold view.

- Select one of the Export as.. commands in the File menu.
- Type in the file name in the standard Save dialog box.
- Import the file into your 2D program.

Exporting 2D DXF files

When you export a 2D DXF file you don't have to worry about the drawing scale and drawing units in Touch-3D™ because these units are not specified in the file. However, when you import the file into your 2D program these things might cause some problems. Many American CAD programs stores the coordinates in inches regardless of what you use on the screen. This means that when you sent a model drawn in millimeters it may come out 25,4 times bigger / smaller than it's supposed to. (1 inch = 25,4 mm) This is usually not very difficult to fix with the built in scaling function found in most drawing programs.

Exporting 2D MiniCad text files

The only thing you have to worry about when you communicate with MiniCad is to use version 2,0 or later and to make sure to use the same units in both ends. You can't use fractional units in Touch-3D™. You have to use one of the decimal types of drawing units while communicating with Touch-3D™.

Exporting 2D EPSF files

When you export EPSF files there are a few things to consider.

- You have to specify the drawing units used in the model before you export it, in order to get a correct scale when you import it into your illustration or desktop publishing program.
- Many illustration and desktop publishing programs lack a drawing scale function. If you want to import a big model, for example a car, you have to specify a scale factor to make it fit into the drawing surface. A car with a length of 4000 mm can for example be scaled down to a length of 200 mm in the illustration program by setting the drawing scale in Touch-3D™ to 1:20 (4000 mm / 20 = 200 mm)
- There are two kinds of EPSF files. The first, called Printer EPSF, is used for placing images into desktop publishing programs. (The command is usually called "Place") This type of file can not be edited. The second type, called Illustrator EPSF, can be opened by many objects oriented Illustration programs such as Adobe Illustrator, Aldus Freehand, Canvas, etc. In this case you are able to edit the model, add surface colors and so on. Many pixel based programs, like Adobe PhotoShop, can also read these files.

Import / export from MS/DOS

The DXF and EPSF file formats are both text based. This means that these files can easily be exchanged between MS/DOS, Windows, Windows NT, OS/2 and Macintosh. The only limitation is that you need a Mac with a 1,4 Meg Superdrive (disk drive). The older 400 and 800 K disk drives can't handle the conversion.

The System 7 package includes an exchange program which enables you to convert files between these platforms. In this case you have to start and run a program before you can convert them. This may be acceptable if you seldom convert files.

If you frequently convert files, it may be a good idea to consider some other software solution where you are able to convert the files more easily. There are a number of separate packages (e.g. Macintosh PC exchange, Soft PC, Access PC etc.) that handles this very well. These programs allow you to handle DOS files as if they were Mac files. The files are converted automatically.

Create your own files

This section is for those of you that write your own programs or want to be able to import file types not supported by Touch-3D™. For this reason we have created a small sample model consisting of two 3D polygons. The model looks like this:

3D MiniCad text files

This shows the syntax for MiniCad text:

The file:	Comments:
UNITS(2,1000);	Copy this (2=mm,3=cm,4=inch,5=m),(no of decimals)
ABSOLUTE;	Copy this
CLOSEPOLY;	Copy this (Indicates a closed polygon)
BEGINGROUP;	Copy this (Start of group of 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)
ENDGROUP;	Copy this (End of group of polygons)

Note that it's very important to copy the syntax accurately. Parentheses, commas and semicolons must be placed in the right places. Note that the period (.) places the decimal point whereas the comma (,) indicates the end of a value.

3D DXF files

This shows the syntax for a DXF-file:

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
0	This command starts the first polygon
3DFACE	Command for a closed polygon
8	Copy this
UNTITLED-1	Copy this (Layer info)
10	Copy this (Code for first X value)
250.000	First X value
20	Copy this (Code for first Y value)
500.000	First Y value
30	Copy this (Code for first Z value)
200.000	First Z value
11	Copy this (Code for second X value)
800.000	Second X value
21	Copy this (Code for second Y value)
1150.000	Second Y value
31	Copy this (Code for second Z value)
250.000	Second Z value
12	Copy this (Code for third X value)
3450.000	Third X value
22	Copy this (Code for third Y value)
1100.000	Third Y value
32	Copy this (Code for third Z value)
100.000	Third Z value
13	Copy this (Code for fourth X value)
3750.000	Fourth X value
23	Copy this (Code for fourth Y value)
300.000	Fourth Y value
33	Copy this (Code for fourth Z value)
-100.000	Fourth Z value
0	End of polygon
0	Copy this (closing commands)
ENDSEC	Copy this (closing commands)
0	Copy this (closing commands)
EOF	Copy this (closing commands)

Note that it's very important to copy the syntax accurately. If you do not follow the syntax carefully the file might come out of phase which will make it unreadable.

Spreadsheet filters

The Spreadsheet import/export filters allow you to communicate with most spreadsheet, database, and word processing programs in a very simple way. You could for example use it for transferring measuring data (coordinates) directly into Touch-3D™. Another possibility is to use your spreadsheet program to perform certain calculations and then transfer them to Touch-3D™ for creation of a visual model.

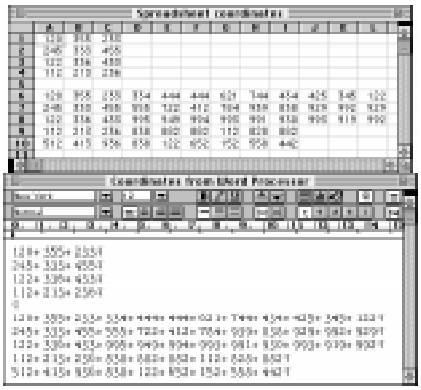
The Spreadsheet filters can be used in two different ways, for importing Locus points and for importing polygons.

Spreadsheet export.

When you import Locus points you simply type in the X, Y, Z values in their respective fields, using one row for each element. If you instead want to import a polygon you need to have three or four sets of X-Y-Z coordinates per row. The two windows at the top of the picture show what an export file looks like in Microsoft Excel. When you export to Touch-3D™, be sure to export the file as a TEXT file, and in TAB format if you have such an option.

Word processor export.

When you export from a word processor you need to separate the coordinates by TABS (using the TAB key). This is not necessary when you export from a spreadsheet or data base program. The two lower pictures show what it looks like in Microsoft Word. Note that we have used the ability to show hidden characters. This makes it easier to check the syntax in the word processor. When you export to Touch-3D™, be sure to export the file as a TEXT file.



Tutorial

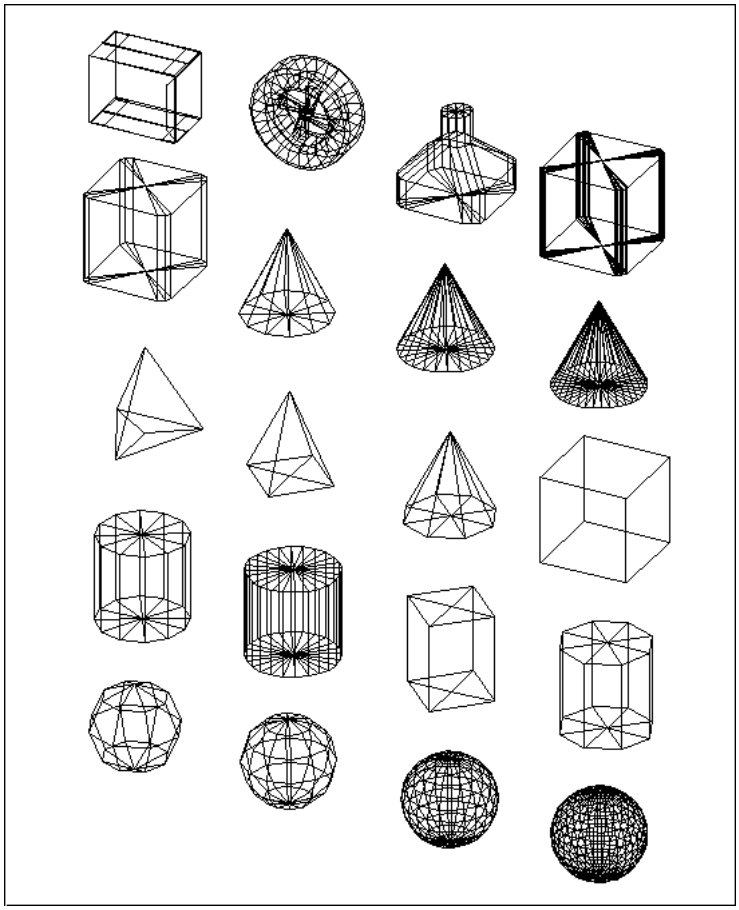
The tutorial section of this manual contains useful tips and hints about how to create your own models, how to resort the unfolding order in order to avoid overlaps, detailed information about import and export, model building, and so on. We hope that it will give you some useful ideas, which you can develop further to suite your own use.

Some of the models are easy to create whereas others are more difficult and require practice. It's, for example, one thing to understand how to model a car, and another thing to actually do it (and make it look good).

Using Shapes

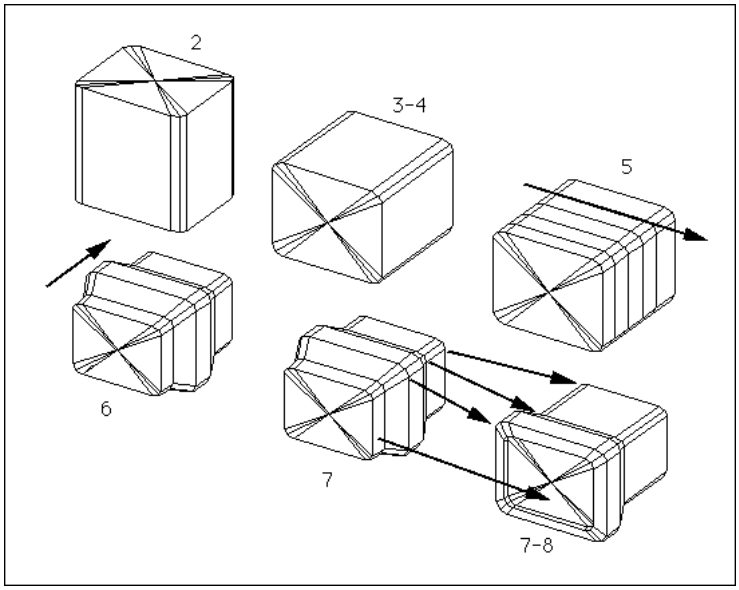
Shapes are used for adding a pre-fabricated component from a supply of pre-fabricated components to drawings, and as a base for creating new, more complex shapes. A shape can be stored in any compatible file format. To import a shape into the currently active drawing, select either the Place..., or Place Import -> commands (File menu).

Below are some examples of shapes included in the package.



Monitor

Below is an example of how a monitor may be created using a shape called RoundCube2D-12, found in the Shapes folder.



1/ Make sure that the Selection mode is set to Mesh mode on the Mode bar.

2/ Import the shape RoundCube2D-12 into an empty drawing.

3/ Rotate the model, seen in the Side view, so that the corner radius can be seen in the Front view.

4/ Select the entire model. Use the Reshape command (Edit menu) to specify the size to for example (X)=500 x (Y)=400 x (Z)=500 mm.

Note that the corner radius can be specified individually by selecting all vertexes of a corner, one at the time, using the Reshape command. The Reshape command is used for defining the dimensions of box, enclosing any number of the vertexes or polygons of a model.

5/ Change to the Top view. Select the entire model. Use the Cut tool (Tool palette) to cut off the model four times horizontally, seen in the top. It's not essential where you place the cuts.

Note that you can lock the angle of the cutting line to horizontal by pressing the Shift key while cutting.

6/ Resize each section, one at a time, to the following dimensions using the Reshape command (Edit menu), as seen in the Top view. Reshape from the center point.

1/ X = 400; Y = 300; Z = 0
2/ X = 410; Y = 310; Z = 0
3/ X = 500; Y = 400; Z = 0
4/ X = 500; Y = 400; Z = 0
5/ X = 400; Y = 300; Z = 0
6/ X = 400; Y = 300; Z = 0

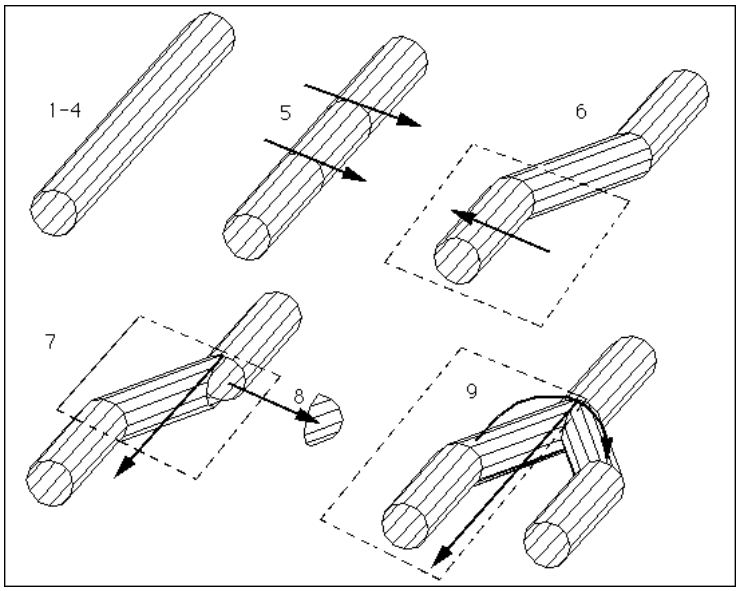
7/ Drag the sections, one by one, to a location similar to the picture.

8/ Look at the finished monitor in the Perspective view.

*The sections where originally located according to the order in which they are connected to one another, starting with the glass surface at the **front** of the model, and ending with the rear of the monitor. In the final model, the glass surface is physically located **behind** the front side of the monitor.*

Y-tube

Below is an example how to create a Y-tube based on a cylinder shape.



1/ Import a cylinder shape from the Shape folder, in this case "Cylinder12".

2/ Rotate it to a horizontal location.

3/ Use the Reshape command (Edit menu) to set the length = 200 mm, and diameter = 20 mm.

4/ Delete the caps at both ends (we want to have a tube, not a cylinder).

5/ Cut it twice using the Cut tool (Tool palette).

6/ Drag two of the sections sideways (Mesh selection mode).

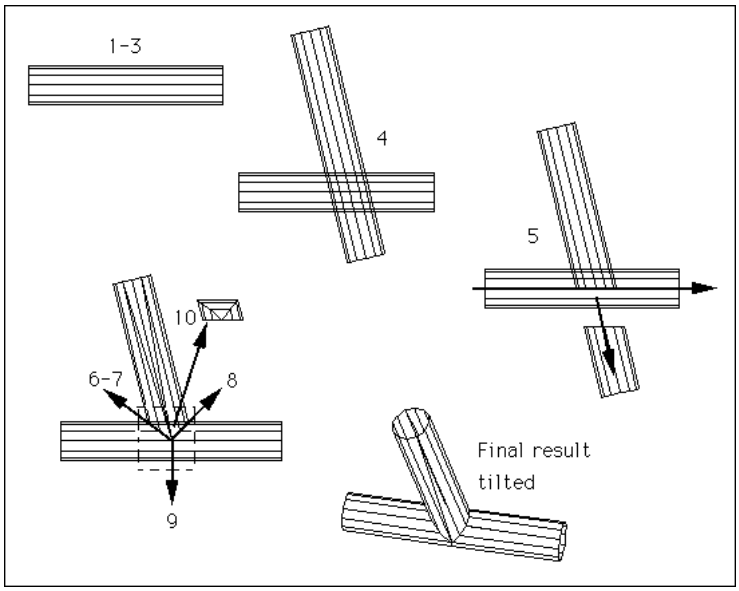
7/ Select two of the sections and use the Cut tool (Tool palette) to cut them off along the center line of the original tube.

8/ Delete the excessive parts

9/ Select the remaining parts. Use the Mirror Copy tool (Tool palette) to complete the model.

T-tube

Below is an example of how to draw a T-tube using a Cylinder shape. The tubes can meet at any angle. The method works best when both tubes have the same diameter.



1/ Import a cylinder from the Shape folder (in this case the Cylinder 24 shape).

2/ Rotate the cylinder to a horizontal position, seen in the Front view, and delete the end caps.

3/ Use the Reshape command (Edit menu) to set the length to 100 mm and the diameter to 20 mm.

4/ Use the Duplicate command (Edit menu) to copy the tube. Rotate it to an angle similar to the one shown in the illustration. Note that they have to overlap one another completely.

5/ Select the standing tube and use the Cut tool (Tool palette) to cut it off along the center line of the horizontal tube. Delete the remaining polygons of the standing tube below the center line of the horizontal tube.

6/ Draw a 2D help line, using the 2D Help Line tool (Tool palette), from the center line of the standing tube to a point above where the outlines of the two tubes meet on one side. Zoom in, and adjust the line so that it runs through the intersection point of the two tubes (outer contours).

7/ Select both tubes. Use the Cut tool (Tool palette) to cut off the tubes along the 2D help line.

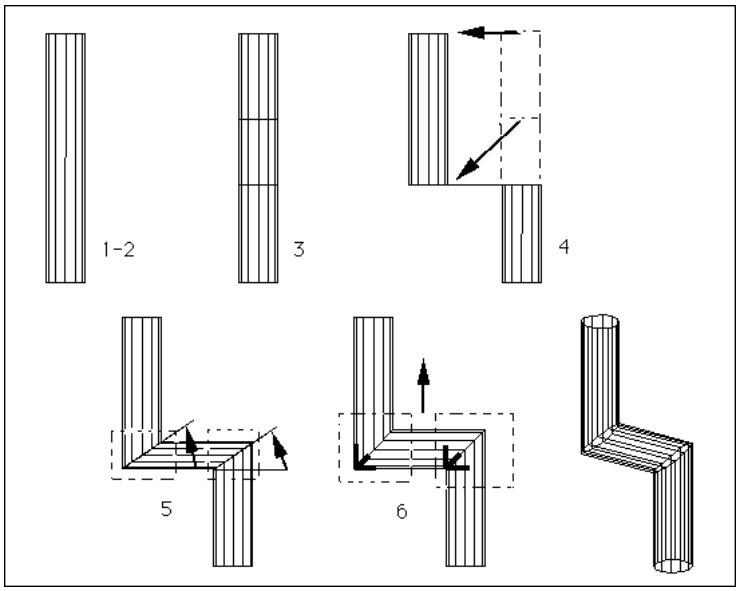
8/ Repeat the steps 6 and 7 for the other side of the standing tube.

9/ Cut the horizontal tube downwards from the center line. The cutting line should start at the center of the vertical tube.

10/ Draw a selection cube around the overlapping parts of the tubes, and press the Delete key to delete the excessive parts.

Z-Tube

Below is an example of how to create a Z-shaped tube using a Cylinder Shape.



1/. Import a cylinder from the Shapes folder.

2/. Reshape it to a length of 250 mm and a diameter of 50 mm.

3/. Add two sections using the Cut tool (Tool palette).

4/. Drag the upper half to the left and downwards to align the two center sections (Mesh selection mode). The horizontal alignment is achieved by activating the Alignment Indicator, in the Preferences command (Page Menu), or by pressing the C key.

5/. Rotate the center sections, one at time, using the Rotate tool (Tool palette), with the rotation mode set to Vertical Constrain in the Mode bar. Be sure to place the center of rotation point at either the far left, or right point of the section when you rotate.

6/. Select one section at the time, and set the width and height to the same value, using the Reshape tool in the Edit menu (reshape from the selected point). This ensures that the cutting angle is set to 45°.

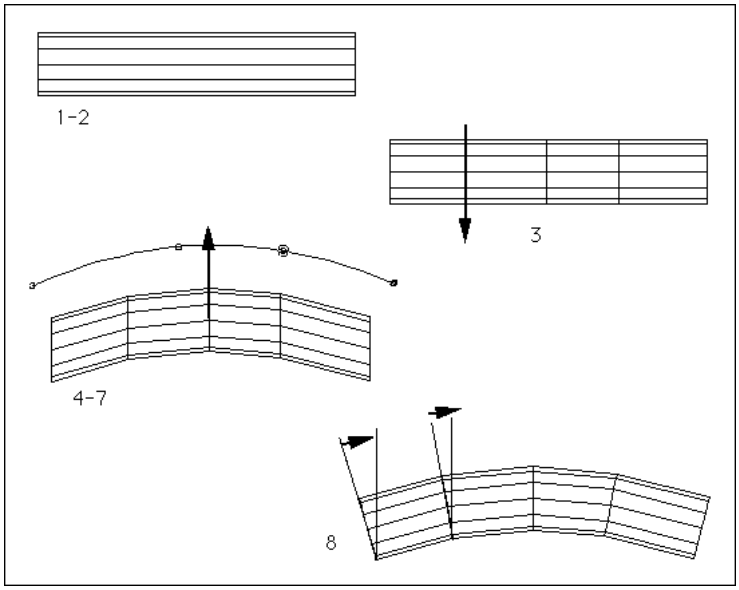
When using the Reshape command in this case it's important to place the Intersection point either to the far left or right vertex of the group, and reshape from the Intersection point in the dialog box. This minimizes the work needed to clean up the model afterwards.

7/. If necessary, adjust the alignments.

Bending a shape

Below is an example of how to bend a shape, using the Alignment Ruler tools to create a smooth extruded curve.

This technique is particularly useful for bending objects containing many cross sections in smooth curves. A technique using manual movements, section by section, is better when the number of cross sections are few, and where the intended bending curve is rough.



1/. Import a Cylinder shape.

2/. Reshape it, using a cross section 50x50 mm and a length of 250 mm.

3/. Add three cross sections, using the Cut tool (Tool palette).

4/. Select the entire shape.

5/. Create a vertical ruler by double clicking on the Vertical Alignment Ruler tool (Tool palette). Type in a name. Place the ruler above the model by clicking out control points. Use three to four points, and make the ruler roughly horizontal.

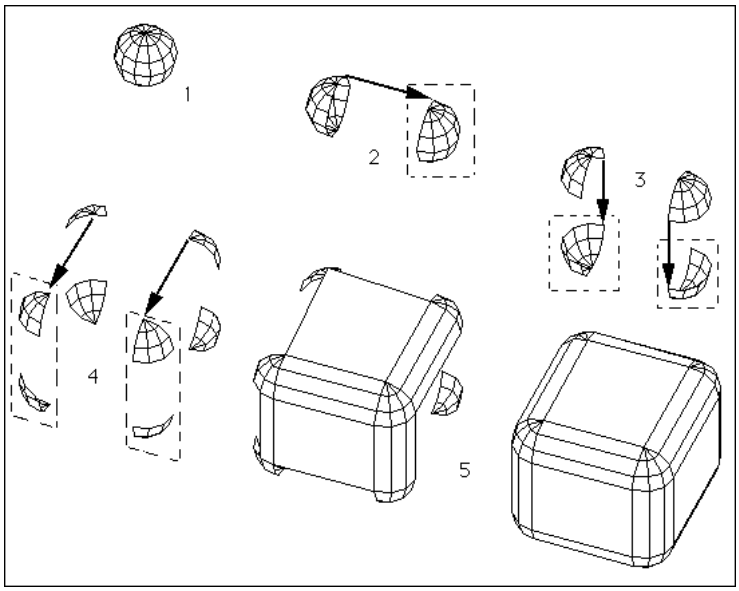
6/. Select the Lock to Rule command (Tool menu).

7/. Drag some of the control points upwards to bend the model.

8/. Use the Rotate tool (Tool menu, Normal rotation mode in the Mode bar) to rotate each section. The angle should be roughly perpendicular to the average angle of the elements on each side of the cross section.
This step is done to reduce the distortion caused by bending the model. If the angle is very steep the cross sections may become very flat.

Rounded cube

Below is an example where a sphere is used for creating a rounded cube.



1/. Import the Sphere 12 shape into the drawing.

2/. Divide the model into two separate components by dragging them apart (Selection mode set to Polygon mode).

3/. Divide the model into four separate components by dragging them apart (Selection mode set to Polygon mode).

4/. Divide the model into eight separate components by dragging them apart (Selection mode set to Polygon mode).

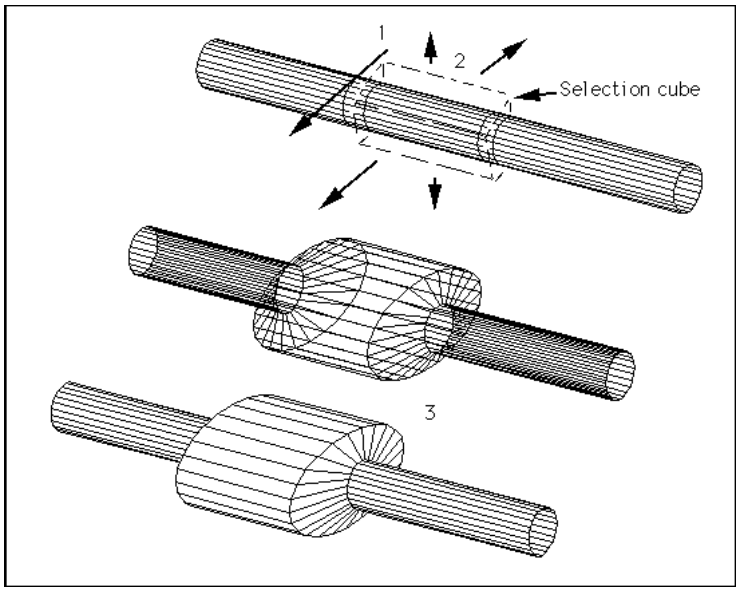
5/. Use the Free Form Polygon tool (Tool palette) to add surface elements between one side and corner of the cube.

A quicker way to add the polygons, after the first corner is done, is to use the Duplicate command (Edit menu) to create copies for the other sides of the cube. Drag and rotate the elements to their respective location using the Selection tool (Tool palette), and the Rotate Left/Right commands (Edit menu).

Note that, in some cases, it's easier to select and drag objects in the Perspective view.

Non uniform cross sections

Below is an example showing how to create a shape with cross sections having varying dimensions. Each cross section can have its own dimensions.



1/. Import the Cylinder 24 shape from the Shapes folder. Add four cross sections using the Cut tool (Tool palette). Delete the end caps.

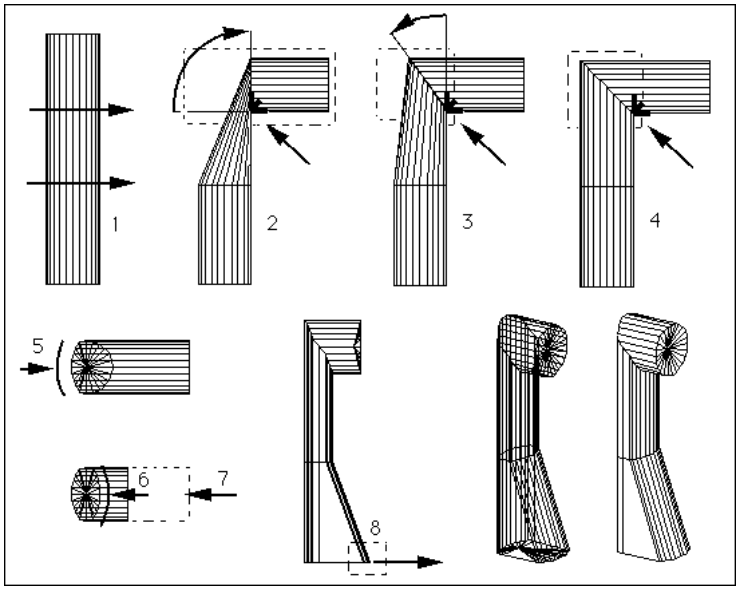
2/. Set the Selection tool mode to Mesh mode (Mode bar), and draw a selection cube around the two sections in the middle, using the Selection tool (Tool palette). Select the Reshape command (Edit menu), and enter a new width and height for the selected vertexes. Make sure to select the center point to scale from before pressing OK.

3/. This is the result. To create a perpendicular connection wall between the wide and narrow parts of the tube, drag the cross sections so that they overlap in the lengthwise direction.

Note that the wide part of the cross section has a bigger width than height. This was achieved by entering different values for width and height in the Reshape dialog box.

Phone

This is a more sophisticated example of how to use a shape as a starting point to create a model. It illustrates the use of a combination of tools to deform and sculpt the original cylinder into a phone (however without buttons).



1/. Import the Cylinder 24 shape into the drawing. Set the dimensions to 50 x 50 x 250 mm, using the Reshape command (Edit menu). Cut it twice using the Cut tool (Tool palette).

2/. Select the upper two sections of the model (Mesh mode in the Mode bar). Rotate the selected parts of the model using the Rotate tool (Tool palette, Normal mode on the Mode bar). Note that it's important to place the center of rotation at the bottom right point of the selected vertexes (where the lower arrow points). Don't forget to lock the rotation angle pointing upwards by pressing the Shift key before clicking to end the rotation sequence.

3/. Select the third section from the bottom, and rotate it back to 45° (Horizontal Constrain mode in the Mode bar).

4/. Select the third section from the bottom (Mesh mode in the Mode bar). Make sure that the Intersection Point is located at the bottom right vertex of the selected vertexes (if not click on it to select it). Select the Reshape command (Edit menu), and enter the same width value as the height value. Make sure to select scaling from the Intersection point before clicking OK.

5/. Select the Top view. Select the rear end of the phone using the Selection tool (Tool palette, Mesh mode on the Mode bar). Create a vertical alignment ruler by double-clicking on the Vertical Alignment tool (Tool palette). Name it and press OK. Click out a ruler that roughly aligns with the back of the phone. Select the Lock To Rulers command (Tool menu). Drag the center part of the ruler to the right to flatten the rear side of the phone.

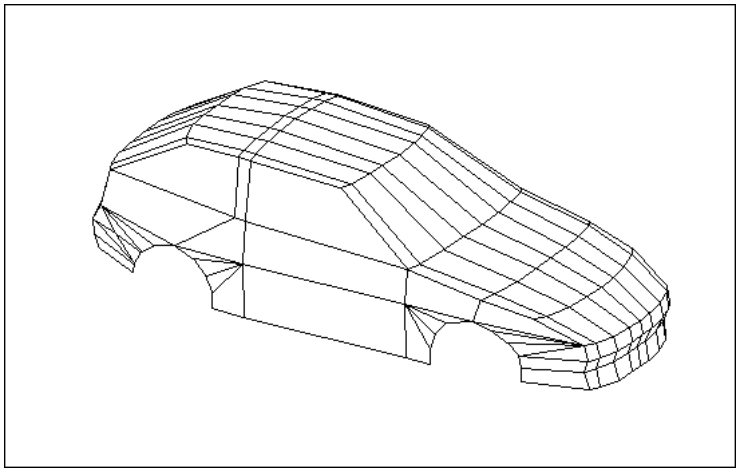
6/. Repeat the actions on Step 5 for the front side of the phone (exclude the top part)

7/. Drag the end of the top part to the left to make it shorter.

8/. Draw a selection cube around the bottom front part of the phone, and drag it to the right to increase the width of the footprint.

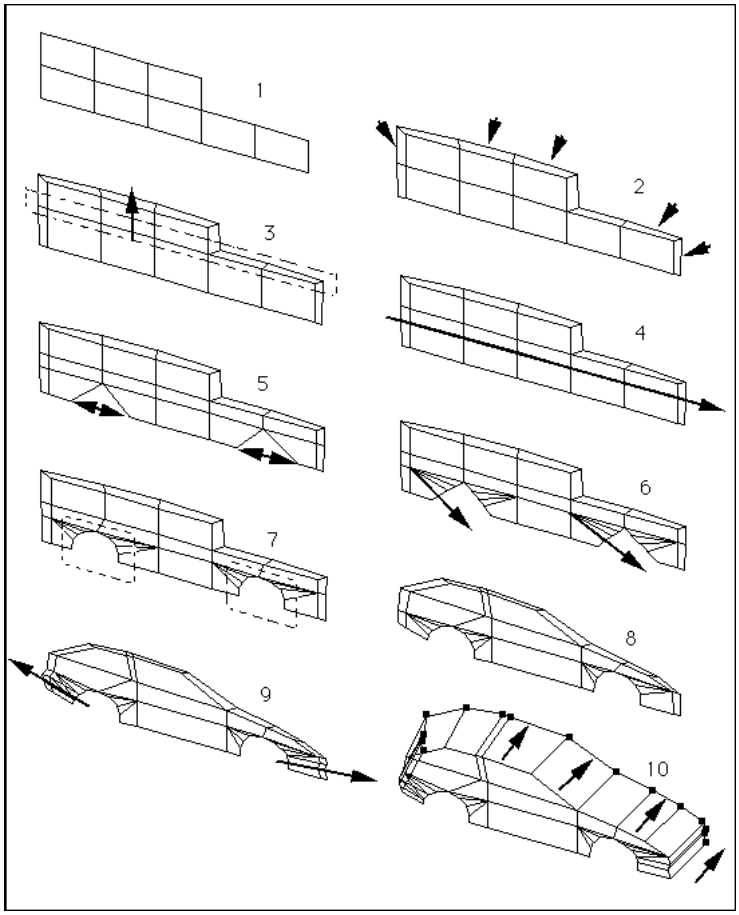
Car body

The car body shown below illustrates a useful technique for free form modelling, where the model has a complex non uniform organic shape.



At the beginning of the process, the model consists of very few polygons, providing the basic outline and shape of the model. The resolution of the model is then gradually increased, primarily using the Cut tool (Tool palette), and free form editing / sculpting. An advantage with this method is that the models tend to be much smaller (using fewer polygons) than models created in other programs. You exercise an absolute control over how the model is divided into sections. A fine polygon mesh can be used where the curvature changes quickly, whereas comparatively flat areas can be represented by fewer polygons.

The level of refinement depends on what the model is intended to be used for. A general rule is to use as few polygons as possible. Excessively complicated models are more difficult to assemble as scale models. A sketch used for scale model building can be fairly rough. A more final model needs more polygons. A model to be used for rendering in a rendering program typically needs more polygons than a scale model to render well (typically requiring an angle between polygons less than 30°).



1/. In the Front view, create two rows of polygons using the Polygon tool (Tool palette). Use the Rectangle mode for the first polygon and Radial Extrusion mode for the remaining polygons. Use the Reshape command (Edit menu) to set the size to 4000 x 1200 x 0 mm (note that the model is still a flat model without depth).

2/. Add a string of polygons surrounding the model along the left, top and right sides, using the Free Form Polygon tool (Tool palette). These polygons will later on be used to form the back, top, and front sides of the model.

3/. Drag the center part of the model upwards slightly to provide space for the wheel housings.

4/. Cut the model using the Cut tool (Tool palette) to divide the lover part of the model in two rows of polygons.

5/. Drag the two vertexes apart as seen on the illustration using the Selection tool (Vertex mode). The openings will later on form the wheel housings.

6/. Use the Cut tool (Tool palette) to cut each side three times as seen in the illustration. This will provide the extra vertexes required to create a roughly rounded opening for each wheel.

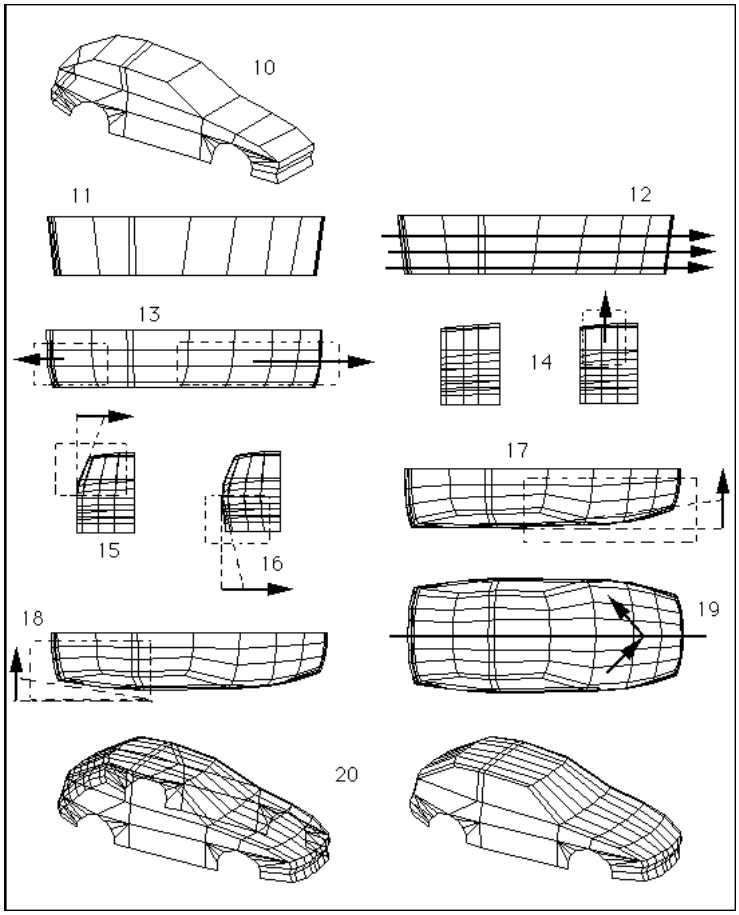
7/. Use the Reshape command (Edit menu) to reshape the opening to a suitable size. In this case it's best to set the Intersection point to either the front or back bottom vertexes for each opening, and reshape from that point. In this way, the bottom vertexes are not moved up- or downwards.

8/. Reshape the model using the Selection tool (Tool palette, Mesh mode in the Mode bar.). Note that the model in the picture looks very different from the previous pictures, but contains in fact the same number of polygons as before. The vertexes have just been dragged around to create a more pleasing appearance.

9/. Add two cutting lines, using the Cut tool (Tool palette), to add bumpers at the front and rear of the car. Adjust the lines to make it look better.

10/. Select all the outermost vertexes as seen in the illustration, change to Top view, and drag them away from the rest to create the width of the car. The model is now a fully three dimensional model, even though it still lacks one side. It's often best to draw only one side, and then mirror copy it to the other side when the model is finished. In this way the model will always be symmetrical.

Select all vertexes and use the Reshape command (Edit menu) to set the size of the body to for example 4000 x 1200 x 850 mm. (850 mm is the width of half the car, meaning that the final width will be 1700 mm when mirror copied).



10/. The same model as shown in the pervious illustration.

11/. Change to Top view.

12/. Select the entire model and use the Cut tool (Tool palette) to cut the model three times as shown in the illustration. Note that one cut should be located fairly close to the side of the car (downwards, as seen in this view).

13/. Change to Top view. Select the front and back center vertexes as seen in the illustration (one at a time), and drag them forwards/backwards. Adjust the vertexes to provide a smooth curve.

14/. Repeat step 13, seen from the front of the body to adjust the curvature of the roof.

15/. Most car bodies have their widest point in the middle part thereof, as seen in the Front view. Draw a selection cube, as seen in the illustration, and use the Rotation tool (Tool palette, Horizontal Constrain mode on the Mode bar) to tilt the roof towards the center of the car. If necessary, adjust the model to create a smooth transition and rounded curves.

16/. Repeat step 15 with the area below the middle part of the body to make the body more narrow at the bottom.

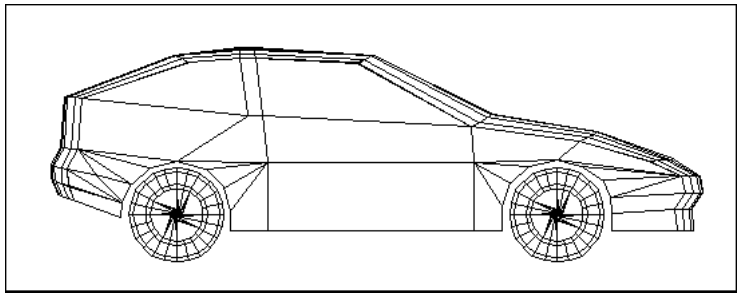
17/. Seen in the Top view, most cars have their widest point somewhere in the middle. In the Top view, select the front end of the body (exclude the center line), and use the Rotate tool (Tool palette, Vertical Constrain mode on the Mode bar) to make the front more narrow. Adjust the model where necessary.

18/. Repeat step 17 for the rear end of the body. Check the model in detail, and adjust where required.

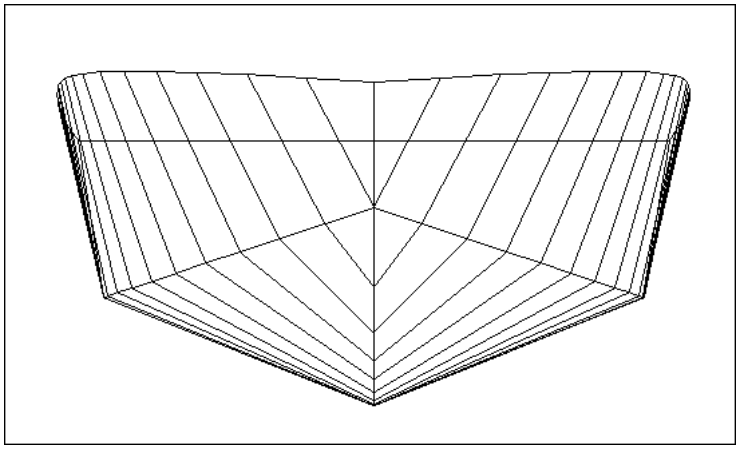
19/. Select the entire model and use the Mirror Copy tool (Tool palette) to draw a mirror line along the center line in order to create a mirror copy.

20/. The model as seen in the Perspective view. If necessary adjust the model, and split it into a suitable number of layers. A well-made model at this level of complexity normally renders smoothly when exported to a rendering program.

It's easier to draw the wheel housings if you insert wheels in the wheel housings. To do so, import a wheel from the Shape folder. Note that the wheel components will automatically be placed in other layers than the active, allowing you to hide the wheels whenever required The illustration below shows the model with wheels.

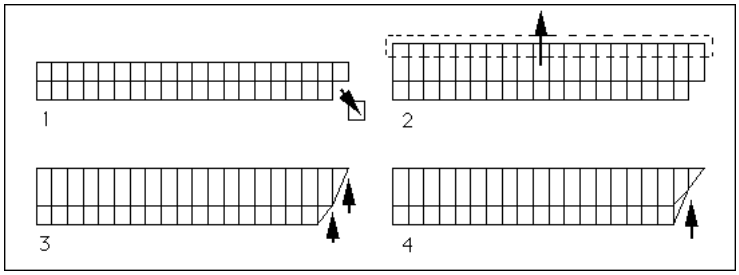


Boat Hull



The boat hull shown above illustrates a technique for free form modelling, using the Alignment Ruler tool.

Some conventions for this model to avoid confusion: The side of the actual boat is here below shown in the Front view with the bow to the right. The Sections are shown in the Side view.



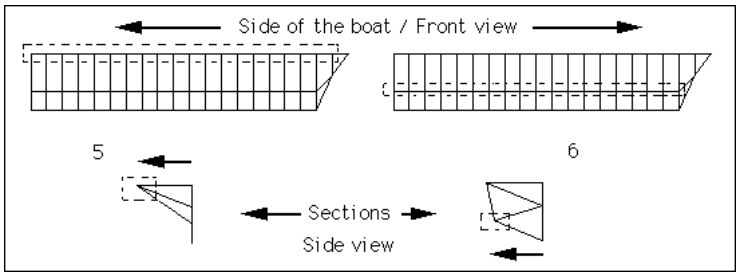
1/. Select the Front view. Create an object mesh to start modelling from by selecting the Create Object Mesh... command (Edit menu). Set the width to 500 mm and the height to 600 mm, choose 2 rows and 20 columns, and set the origin point at X=0, Y=0, Z=0. Click OK. The mesh is being created, having the dimensions X=10000 mm, Y=1200 mm and Z=0 mm. Delete the bottom right polygon. The bottom row will form the bottom of the boat and the upper one will form the side. Note that the bottom left vertex has the coordinates X, Y, Z = 0 which is convenient because it provides a reference point for measuring.

At this point it may be suitable to set the Printing Scale (Page Menu) to 1:50) and use the Move Page command (Page menu) to center the printing frame around the model. Note that this has to be done individually, view for view, to provide an accurate centring of the model on the paper when you change view.

2/. Select (Mesh mode) the upper row of vertexes. Click on the Y coordinate, enter a new Y value (1800 mm), and press the Return key to move the row upwards.

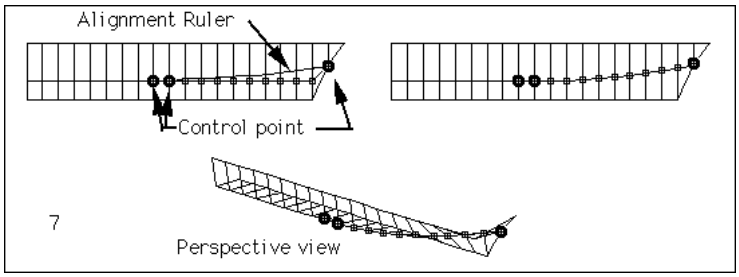
3/. Drag the bottom right vertexes of the two front polygons upwards to create two triangles.

4/. Select the vertex where the arrow points, click on the Y coordinate, type in Y=1100, and press the Return key to move it upwards.



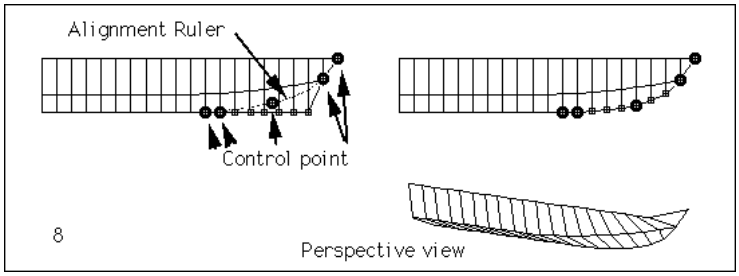
5/. In the next two steps a depth is added to the model. Select all but the front vertex of the top row, click on the Z coordinate and enter the coordinate value -1750. Press the Return key. The reason for entering a negative value is that we wanted to add the depth towards us (check the coordinate system section of this manual for further explanation).

6/. Select all but the front vertex of the center row, click on the Z coordinate and enter the coordinate value -1500. Press the Return key.

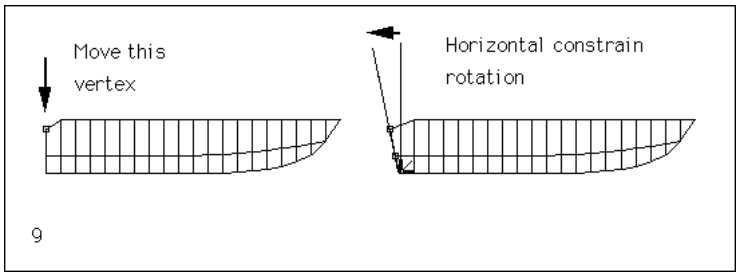


7/. At this step we are going to create a smooth chine curve using the Vertical Alignment Ruler tool (the chine is the joint between the bottom and the side of a boat hull). Select the Front view. Select the same number of vertexes as shown in the

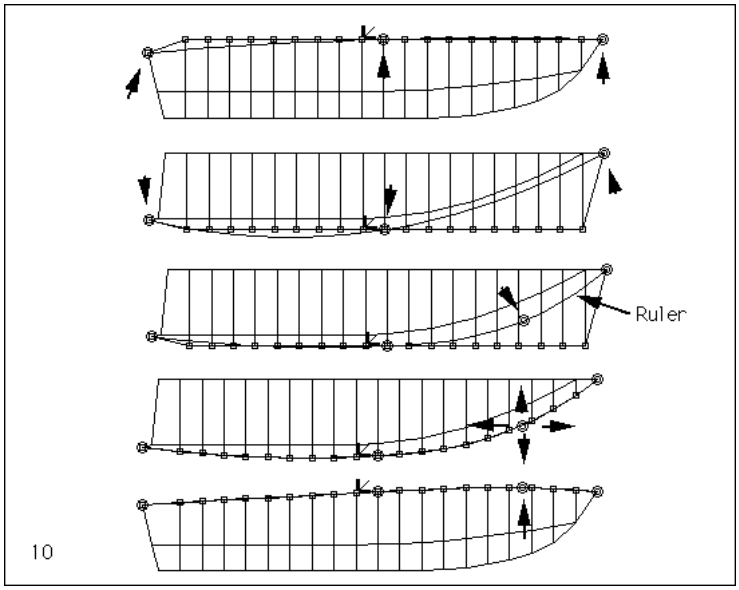
illustration (not the vertex along the center line). Double click on the Vertical Alignment Ruler tool (Tool palette). Type in the name "Chine". Click out three ruler control points as shown in the illustration. Change view to check that the curve looks good in all views. If so, click on the 3D Alignment execution button on the Mode bar to align the vertexes to the Chine ruler. Check the result in all views.



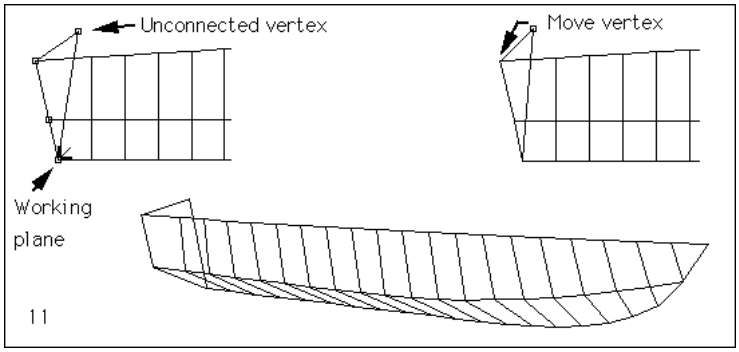
8/. In this step a smooth center line curve is being created, using the Vertical Alignment Ruler tool (Tool palette). Select the Front view. Select the same number of vertexes as shown in the illustration. Double click on the Vertical Alignment Ruler tool (Tool palette). Type in the name "Center line". Click out the ruler control points as shown in the illustration. Note that the first and last two control points are placed on vertexes, whereas the one in the middle is used for controlling the curve. Click on the 2D Alignment execution button on the Mode bar to align the vertexes to the ruler (it's only necessary to align it in 2D because this curve follows the center line). Check the result in all 3D views.



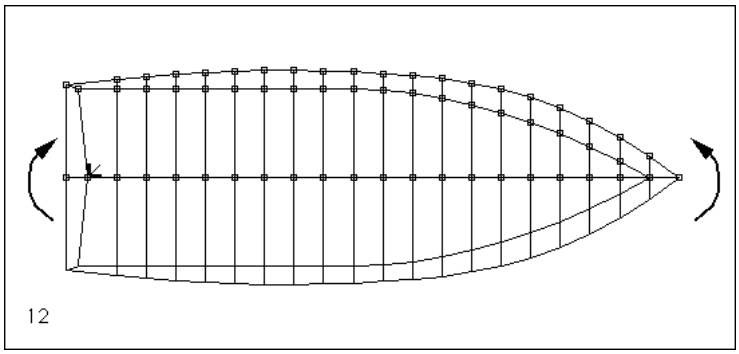
9/. Select the top left vertex in the model. Enter the following new coordinates: X= 0, Y= 1600, Z=-1600. Select the three vertexes at the back of the model, and rotate them backwards, using the Rotate tool (Tool palette) in Horizontal Constrain mode (Mode bar). Use the bottom left vertex as the center of rotation.



10/. At this step a smooth sheer line curve is being created, using the Vertical Alignment Ruler tool (Tool palette) (the sheer line is the joint between the top of a hull and a deck). Select the Front view. Select all vertexes in the top row, except the one in the front which we want to maintain at the present location. Move the Intersection point to a vertex at the center of the model. This will provide an efficient centring when the view is changed. Double click on the Vertical Alignment Ruler tool (Tool palette). Type in the name "Sheer line". Click out the three ruler control points as shown in the illustration. Note that all three control points are in this case placed on vertexes. Change to Top view. Add another control point in the front part of the hull and drag it to make the curve look as seen in the illustration above. Change to Front view. Drag the new control point upwards slightly to make the curve look more smooth. When this is done, click on the 3D Alignment execution button on the Mode bar to align the vertexes to the ruler. Check the result in all 3D views. The hull should now have smooth curves.

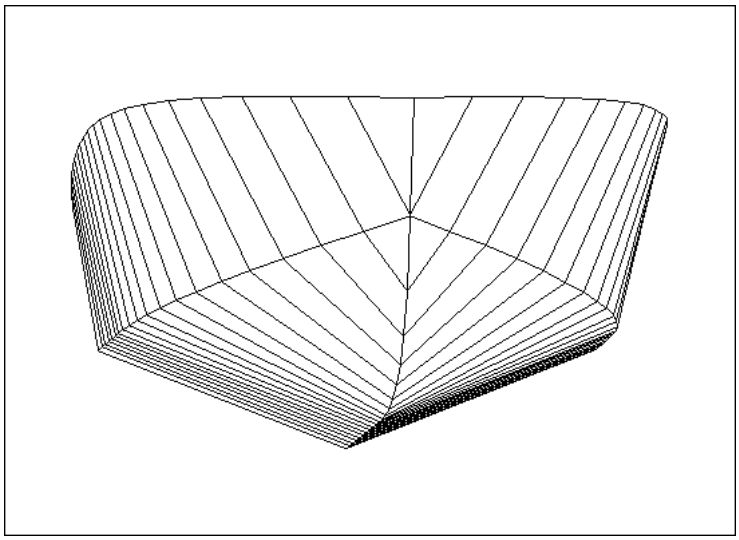


11/. In the next step, the transom (back end) of the hull will be added. Select the Front view, to see the hull from the side. Click on the bottom left vertex of the model to place the Intersection point / working plane at the center of the hull. Draw a polygon as seen to the left in the illustration. Note that one of the vertexes is placed unconnected. This is done because it will then be placed at the working plane (in this case the center line). Select the unconnected vertex, and drag it so that it snaps to the top left end of the model. It will now snap in the X and Y direction, but will remain along the center line in the direction away/towards you. This would not have been the case if the polygon had been drawn with connecting vertexes.



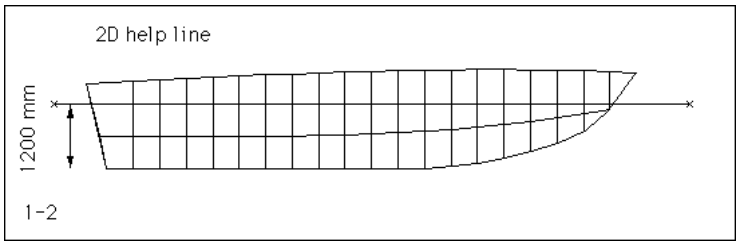
12/. The final step is to mirror copy the hull side to the other side. Select the Top view. Select the entire model. Select the Mirror Copy tool (tool palette), and draw a mirror line along the center line.

The illustration below shows the final model in the Perspective view.



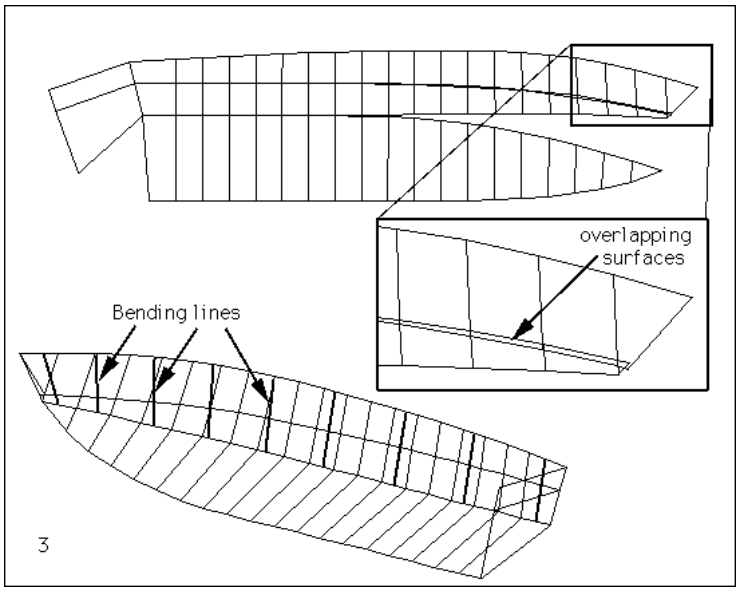
Unfolding curved surfaces

This chapter continues using the model created in the previous chapter. It illustrates a technique for accurately unfolding a complex, twisted shape.



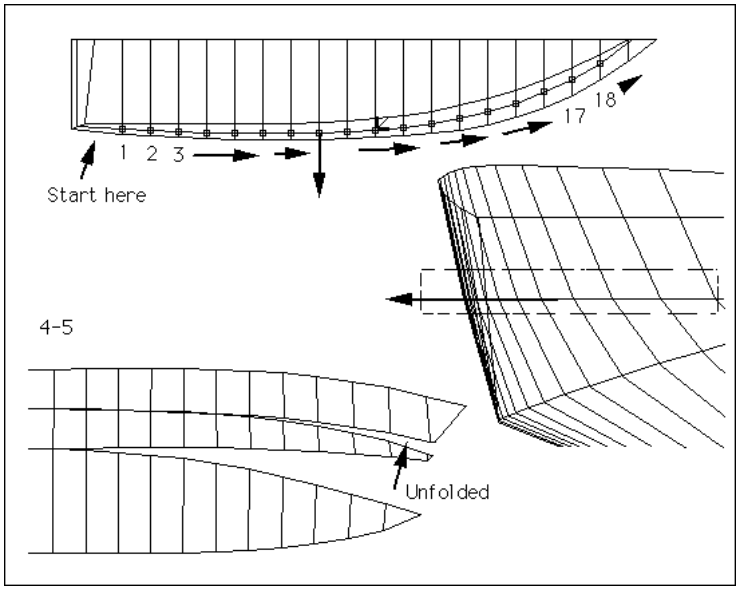
1/. Open the boat hull from the previous example. Delete one of the sides if the hull was saved with two sides.

2/. Select the Front view. Draw a 2D help line by double clicking on the 2D Help Line tool (Tool Palette). Enter the following line coordinates X1= -1000, Y1= 1200, X2= 11000, Y2=1200. Click OK. Select the entire model. Use the Cut tool (Tool palette) to cut the side of the hull along the 2D help line.



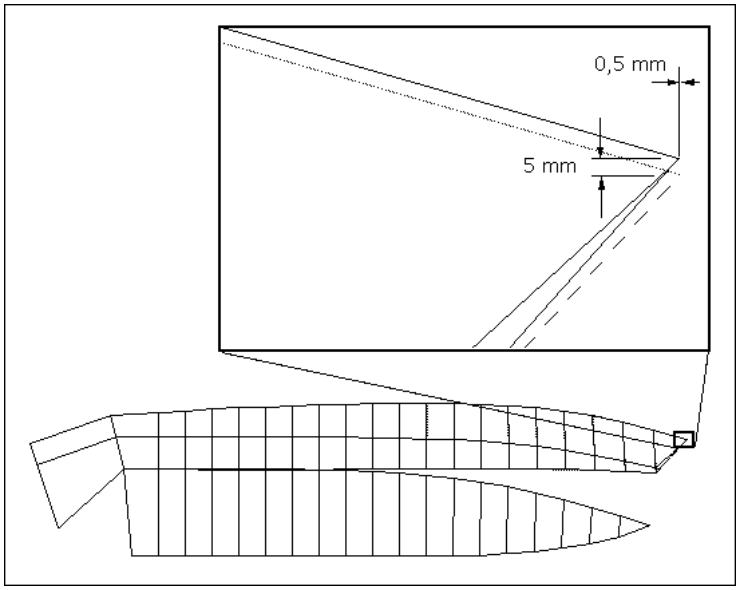
3/. Select the Unfold view. Zoom in on the front part of the hull side. Note that the two rows now overlap. Why? The reason is that the bending lines do not follow the lines of the cross sections. The bending lines are the direction where the lines remain

straight, if you drag a rule over the hull. The thicker lines illustrate their approximate locations along the hull side.



4/. To compensate for the not aligning curves, drag the center part away from the center line of the hull. In this case we have exaggerated the stretching to illustrate what happens.

5/. A more realistic method for doing the job is to start where the two rows are connected, in this case at the back, and gradually move forward (1-18), moving the vertexes slightly away from the center line. The result of a movement can instantly be monitored by clicking on the Unfold view button (Mode bar). The movements are usually very small. In most cases it's best to use the arrow keys for the movements (in this particular example peaking at about 13 mm, and averaging at about 5 mm). Since a boat is a fairly big object, it's practical to set the grid (Set Grid, Page menu) to 1 mm steps. When the two rows align within 1 mm in the Unfold view, the compensation is ready. Why not increase the accuracy even further? Because other error factors tend to be more significant. Such error factors may for example be caused by measuring errors, cutting errors, thermal expansion, handling problems for a sheet of this size, and so on.



6/. How much does the additional line improve the accuracy of the calculation? To check this we have placed the new unfolded pattern over the original. The maximum difference was Y= 5,062 mm and X = 0,541 mm.

Conclusions and some general unfolding tips.

This example illustrates that splitting a panel into two strips can improve the accuracy, but it may not always be necessary. If the unfolded pattern is to be used for a real boat it's advisable to divide the panel into two strips for better accuracy. When used for a scale or sketch model, it's probably a waste of time.

What about dividing it into three strips? It may be recommended if the panel is more curved and twisted. In this example it would probably not add much accuracy (< 1 mm), and would be much more time consuming to do. An example of where it could be worth while is the bottom panel which is more curved and twisted than the side panel, in the bow area.

It's important to understand that additional strips also add internal fluctuations to the model which could result in a lower accuracy if the panel is divided into an excessive number of strips.

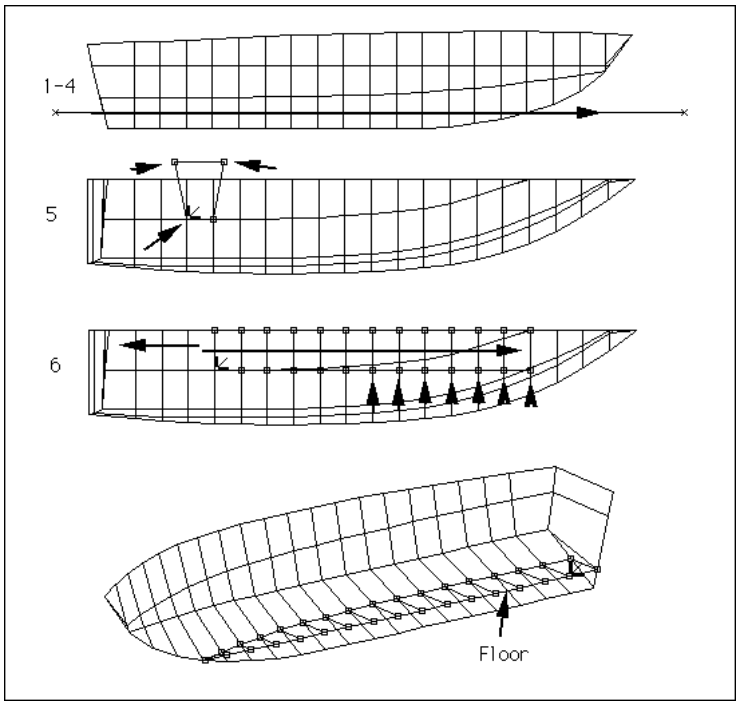
A general rule is to use as few polygons as possible, without loosing too much accuracy.

In the final run, however, it's up to your experience to decide what to do.

Adding an internal structure

Touch-3D™ can of course also be used for drawing an internal structure, for example bulkheads and framing. An advantage when using the surface model as a base for the internal structures is that it allows you to fit any internal structural surfaces to the skin of the external model. In the boat hull, for example, you can calculate the dimensions of floors, bulkheads, and other panels aligning to the model surface.

It's usually easy to fit an internal element to the surface model if there are vertexes to snap to. It's, however, much more difficult to fit in something that doesn't have snap points. Below is an example showing a useful method.



1/. Open the boat hull used in previous chapters.

2/. Create two new layers, named Hull Points and Floor (Organize menu, Layer Configuration...). If necessary, rename the original hull layer to Hull.

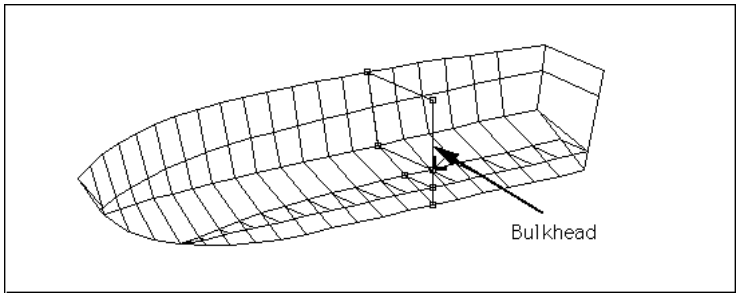
3/. Select the entire hull and copy it (Edit menu, Copy). Make the Hull Points layer active (Organize menu, Active Layer -> Hull Points). Paste the copy into the Hull

Points layer (Edit menu, Paste). We now have a copy of the hull to be used for calculating measuring points on the hull.

4/. We are now going to calculate the shape for a floor, located at a lever 300 mm above the lowest point of the hull, located at Y=0. To do this, select the Front view. Create a 2D help line by double clicking on the 2D Help Line tool (Tool Palette). Enter the following coordinates X1=-1000, Y1= 300, X2=11000, Y2=300). Select the bottom panel, and cut it off along the 2D line using the Cut tool (Tool palette). By cutting off the bottom panel, an intermediate line of measuring points will be formed.

5/. Select the Floor Layer (Organize menu, Active Layer-> Floor). Set the Layer Visibility mode to Show Snap (Organize menu). Select the Top view. Click where the arrow points to place the working plane at Y=300. Draw a polygon using the Free Form Polygon tool (Tool Palette). Note the two vertexes at the intermediate points should be snapped directly to their final locations, whereas the two vertexes along the center line should be placed beside their final locations. The reason for this is that they will then be placed at Y=300 (the working plane). Drag them to their respective location.

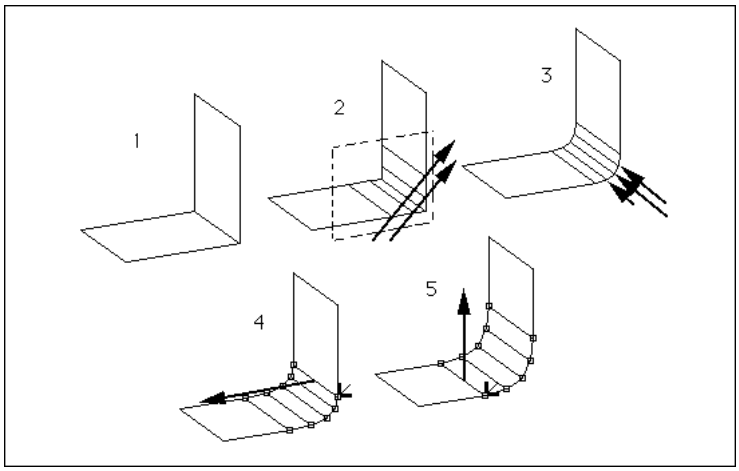
6/. Select the polygon we just created (all four corners). Use the Polygon tool (Tool palette), Radial Extrusion Mode (Mode bar) to extrude the rest of the polygons for the floor panels (forwards and backwards). Adjust the outer side to align to the intermediate points, one by one, by dragging their respective vertexes to that location. We have now created a new floor.



Above is an example showing how a bulkhead has been added to the model, at one of the current sections. The technique is similar to the one described in the Boat Hull chapter (adding a transom).

A corner radius

When a piece of sheet metal is folded in a folding machine, a rounded edge is formed. The radius varies with the thickness of the sheet and type of material. When using polygons, it is assumed that the edge is absolutely sharp, which causes a small calculation error. This small example shows a method for reducing such an error.



1/. Draw two polygons and place them perpendicularly to one another, with connecting sides.

2/. Select a view that shows the section of the two polygons (forming an L-shape). Select both polygons. Draw two (or as many as you like) cutting lines, using the Cut tool (Tool palette), to split the edge part into a number of small polygons.

3/. Drag the vertexes marked inwards to simulate a radius.

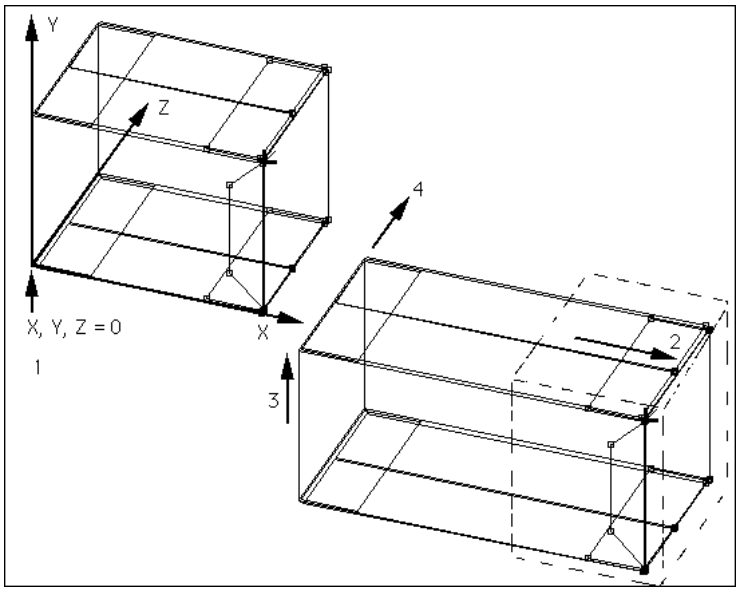
4/. Select all vertexes in the radius. Place the Intersection Point as shown in the illustration. Use the Reshape command (Edit menu) to set the horizontal radius to a suitable value.

5/. Select all vertexes in the radius. Place the Intersection Point as shown in the illustration. Use the Reshape command (Edit menu) to set the vertical radius to a suitable value. Note that the horizontal and vertical dimensions should be the same to create a circular radius.

Cardboard box

Parametric design.

This example illustrates not so much how to model a box, but more how to use Touch-3D™ in a parametrical way. It also illustrates some unique modelling features, such as the ability to add surface elements in the Unfold view and how to use the Cut tool in the Unfold view.



1/. Import the shape Cardboard box from the Shape folder. Note that the left bottom corner of the box is located at X, Y, Z = 0. This makes it easier to adjust the dimensions for the rest of the box.

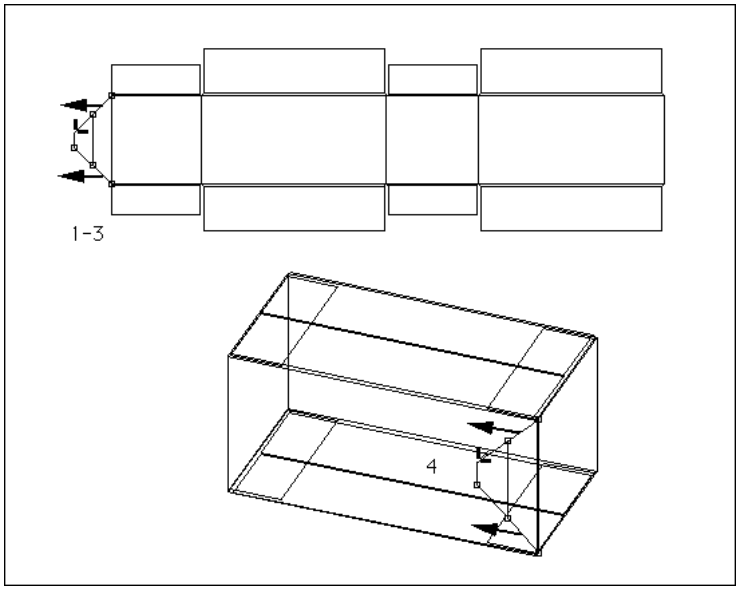
2/. Select the Front view. Select all vertexes on the right side of the box, including the assembly tab. Be sure to place the Intersection point at a vertex as far to the right as possible. Note that there are several overlapping layers of materials in this model which are separated 3 mm to compensate for the thickness of the material (corrugated cardboard). By placing the Intersection point at the right end of the box we get the size of the box, in this case 200 mm. Enter a new width in the X direction, click on the X coordinate (Data Display bar) and enter a new value, for example 300 (mm).

3/. Repeat step two, this time entering a new height (y-value).

4/. Select either the Side or Top view. These views are used for changing the depth of the box. If the Top view is used select the upper side of the box. If the Side view is used, select the right part. As before, make sure to place the Intersection point correctly, and then enter a new Z value.

Adding new elements in the Unfold view.

In a cardboard box design, it may be difficult to add new surface elements to the model, because the model often consists of several overlapping surfaces. For this reason it's often easier to add new components in the Unfold view. Below is an example showing how it can be done.



1/. Click on the Unfold view. Select a polygon using the Polygon selection Mode (Mode bar).

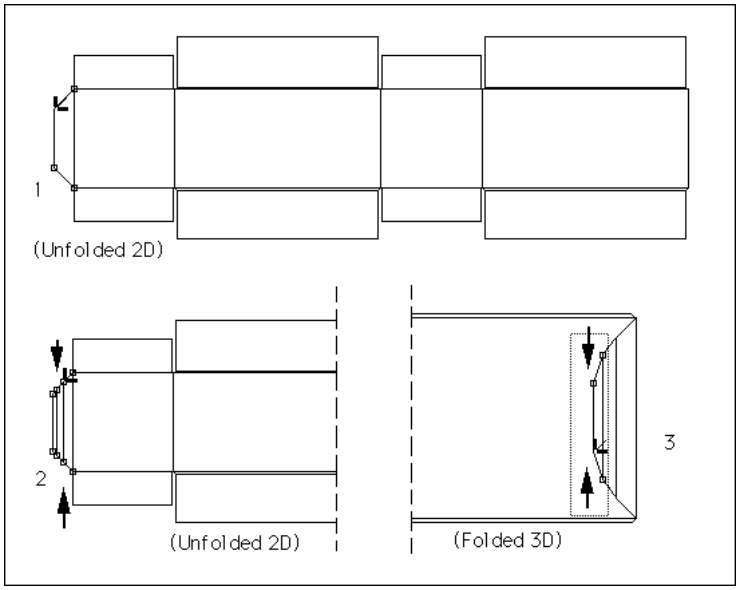
2/. Select the Polygon tool (Tool palette), and set the drawing mode to Radial Extrusion (Mode bar). Add a new surface or surfaces. In this case we have chosen to add a single polygon.

3/. Select the unconnected part of the polygon (Mesh mode, Mode bar).

4/. Change to a 3D view, and drag the two selected vertexes to their final location. Note that the Radial Extrusion tool uses the 3D direction of the polygons from which it was extruded, and stretches it.

Using the Cut tool in the Unfold view.

Another useful method is to use the Cut tool (Tool palette) to cut polygons in the Unfold view. This may in many cases be more understandable than doing it in a 3D view. Below is an illustration showing how to round off an assembly tab.



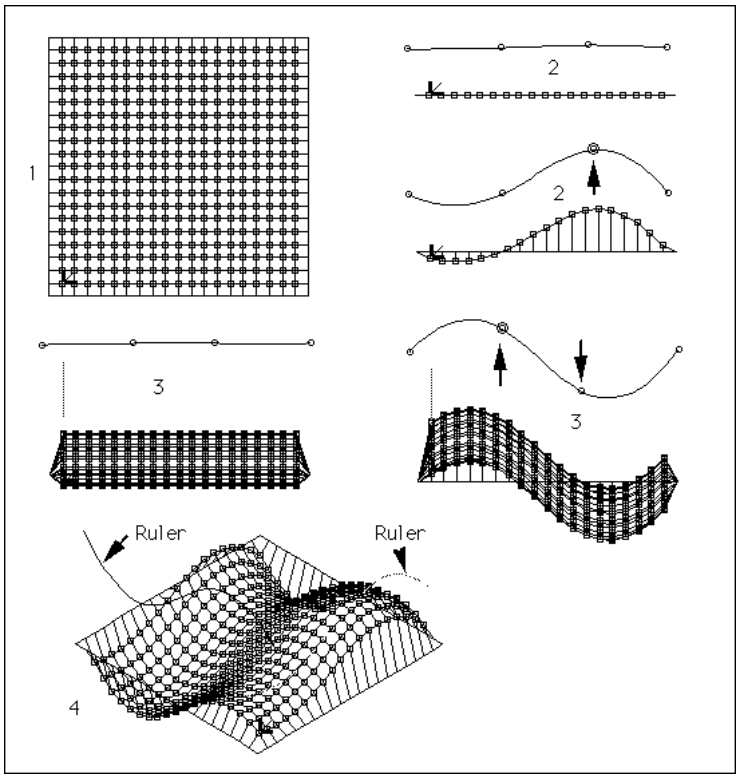
1/. Click on the Unfold view. Select the tab to be rounded, using the Polygons mode (Mode bar).

2/. Select the Cut tool (Tool palette) and cut the tab twice.

3/. Move the new vertexes to create a rounded shape. Note that vertexes can not be moved in the Unfold view. It has to be done in one of the 3D views.

Terrain model

Below is an example showing how to use Touch-3D™ for creating terrain models using multi-directional Alignment rulers.



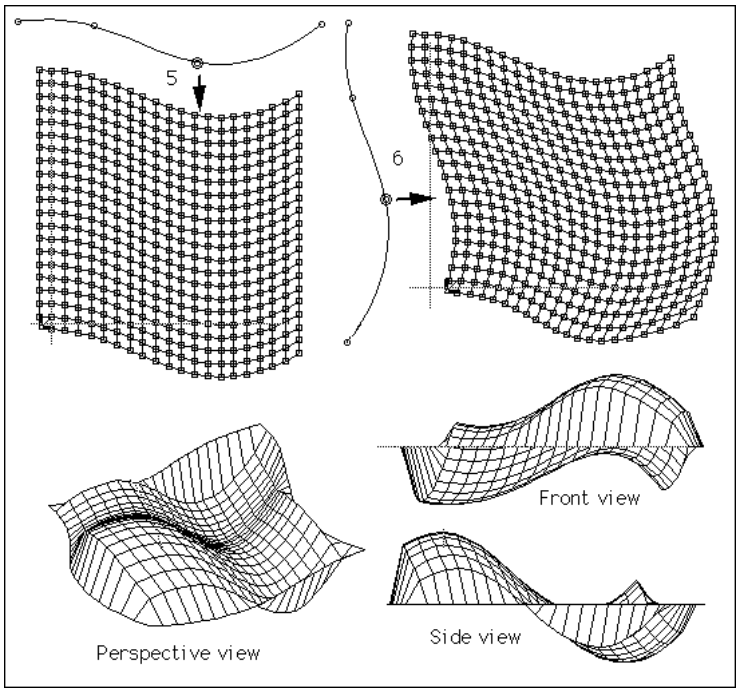
1/. Select the Top view (in this case the model is to be oriented horizontally). Create a mesh using the Create Object Mesh command (Edit menu). In this example the grid consists of 20 by 20 polygons. The size of each polygon is not important, but you may need the adjust the Paper Scale (Page menu) to make the model fit into the drawing surface, and move the paper relative to the model for each view (Move Paper command, Page menu). Select all vertexes (Mesh mode), except the vertexes in the outer rows / columns, which will be left at their original location in the Y direction.

2/. Select the Front view. Double click on the Horizontal Alignment Ruler tool (Tool palette). Name the Ruler. Click out four ruler control points (roughly horizontal). Select the Lock To Ruler command (Tool menu) to turn it on. Select one of the control points and drag it upwards, as shown in the illustration.

3/. Change to the Side view. Double click on the Horizontal Alignment Ruler tool (Tool palette). Create a roughly horizontal ruler. Select one of the control points and drag it vertically, to create a curve.

4/. Select the Perspective view to see the result. Note that the outline of the model has remained at the original location, because these vertexes where not selected.

To further illustrate the ability to deform a model in several directions, using any number of vertexes we will now deform the entire model, seen from above, both horizontally and vertically.



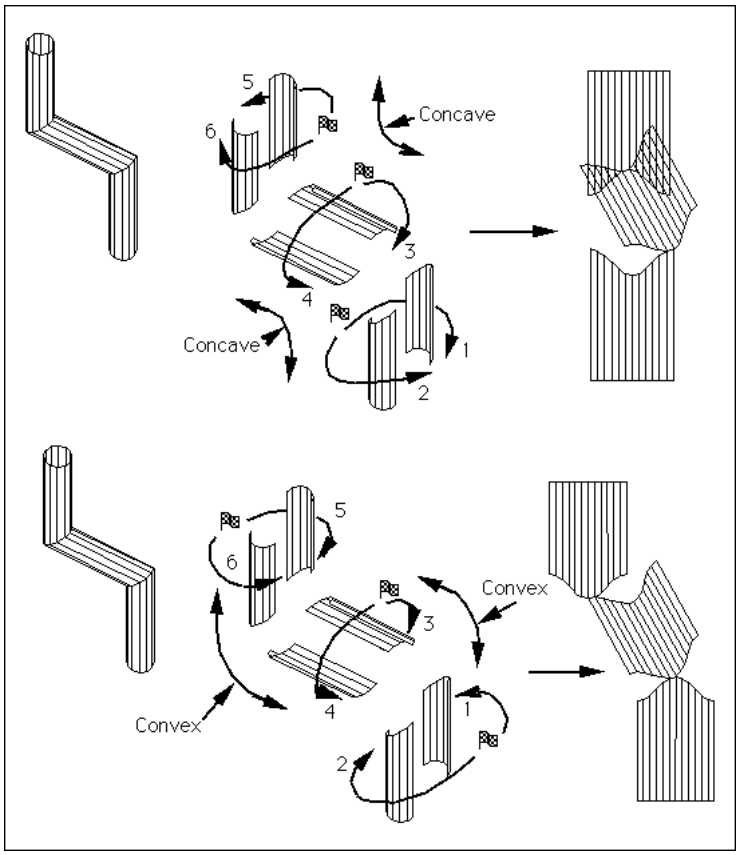
1/. Select the Top view. Select all vertexes (Mesh mode). Double click in the Horizontal Alignment Ruler command (Tool menu). Deform the model vertically.

2/. Double click in the Vertical Alignment Ruler command (Tool menu). Deform the model horizontally.

The result can be seen at the bottom of the illustration. Note that the entire model was deformed in this case. You can of course use as many vertexes and rulers as you like to create local deformation effects.

Some useful unfolding tips

The process of unfolding in an efficient way may seem complicated at first. This chapter contains some practical tips and hints on how to unfold in the best possible way.



Convex and concave surfaces.

As a general rule, it's always best to start an unfolding calculation at the peak of a convex surface. Concave surfaces always tend to create overlapping surfaces. It may not always be possible to do this, especially when a model changes bending form convex to concave surfaces. Below is an example showing how a model can be sorted to avoid overlaps.

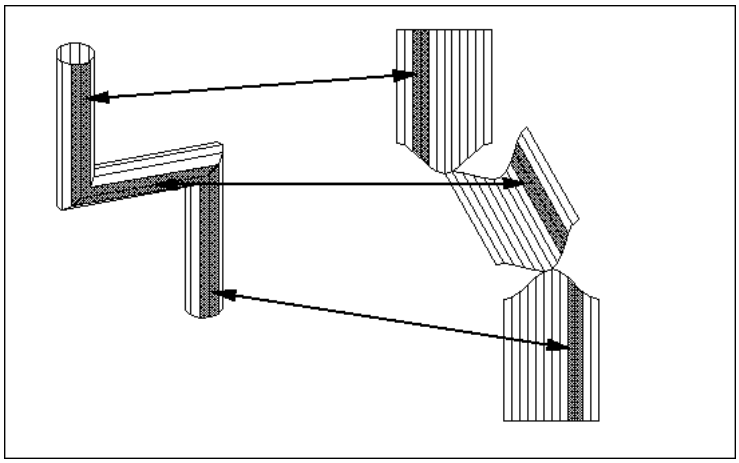
Dividing up models using layers.

Layers can be used for dividing a model into separate components, which will be unfolded as separate units. This can be a good method to quickly bringing order into

a very unorganised model. It's also important to know that layers can be exported separately, which can be useful for finale editing in an external program.

Resorting the model manually.

Touch-3D™ can also be used for reorganizing the unfolding order manually. To do this, export the unfolded model, and import it again into the program. When the model is imported it is placed in 3D in the Front view, and may be edited as any other 3D object. If it's done using the Polygon Selection mode, the shapes of the polygons won't be changed, but they may be dragged and rotated to any angle. This method is also useful for creating a ready to use printing pattern, which makes the most out of the available material.



Using colors.

One way of making it easier to understand and interpret an unfolded pattern is to set the line colors differently for some polygons in a model. It may be either a single polygon, or a row of polygons. The illustration above shows an example.

Printing and building models.

A big advantage with Touch-3D™ is that you can produce the models with almost any kind of 2D printer or plotter.

Plotters.

A pen plotter is useful because it can plot on a comparatively thick material. This means that you may not have to reinforce the surface, just cut out the pieces and fold them. You can also, to some extent, apply colors (usually 8 colors at the time) and surface structure to the model even though these qualities are somewhat limited.

Cutting with plotters

Some types of plotters are able to cut out pieces directly from a thin material by replacing the ordinary pen with a knife. This is a method used for stripes and plastic letters. Touch-3D™ can in this case be useful in several ways.

One example is that you can use a tape instead of painting a surface. Touch-3D™ can in this case help you to calculate the unfolded shape of the object to be “painted”. You then cut out a tape accordingly and apply it on the surface you want to "paint".

In some cases where for example a straight stripe is to be applied on a curved and/or twisted surface you have to modify the “strait” line so that it will look strait when applied to the surface. In reality the “strait” stripe may have to be curved. If the surface is twisted, it's very likely that the stripe may vary in width when you unfold it. Import the stripe into Touch-3D™. Add the estimated twist to it and export it back again as an unfolded shape.

Cutting with NC controlled machines.

There are many types of computer controlled cutting machines available on the market, for cutting out sheet materials; laser, plasma, high pressure water, saws, knifes, milling cutters, etc. These machines are usually controlled by NC software from some sort of computer. This type of software can usually read DXF files, which means that you can easily use almost any of these machines for cutting out your patterns.

Color printers.

Many color printers have a very good color control. You can build very realistic models and mock-ups that includes both color and surface structure. The printer delivers a ready to use surface. You don’t have to spend time painting the model. You just have to cut out the pieces and fold the model.

A color printer can usually not handle thick materials like cardboard. Some sort of reinforcing material has to be added. You can use almost any type of reinforcement material such as cardboard, plastic sheets, plywood, hardboard, etc..

Photographic surfaces.

Another method to build a color model is to use a photographic surface. It's comparatively inexpensive to convert a computer graphics file into a photo. You simply send an EPSF file on a disk to a specialist and they convert the image into a photo. This photo can be enlarged into a substantial size. The advantage is that a photo can show very bright, strong and vibrant colors which you can't achieve in any other way. You can also control the degree of glossiness of the surface by selecting a suitable type of photo paper.

A few practical model building tips.

In this chapter we have included a few hints and tips that we hope will inspire you to use Touch-3D™ in a creative way.

Combinations of programs

It's important to remember that you can use a wide variety of programs in combination with Touch-3D™. You can for example start by defining your model in a CAD program, unfold the model in Touch-3D™, and add surface structures in an illustration program. You can for example scan in a photo (or use Photo CD) of an exterior wall of a house. A real wall may have small variations in color, surface structure, maybe some cracks and so on. This can make a house model look incredible realistic. Don't be afraid to use a series of programs to achieve a good result. *Experiment!*

Glossy surfaces on models

When you use an ordinary printer (laser or ink jet) you usually use ordinary paper which doesn't have a glossy surface. If you want a glossy surface you can print the model on overhead film. Print a mirrored version of the image on the back side of the film. Since the overhead film is transparent, you have to add a white surface at the back of the film (the same side as you printed on). The best way to do this is to use the back side of a white tape or gummed paper. In this way you will get a sheet with a very glossy surface and where the ink is protected and enclosed.

Another advantage with this method is that when you use ink jet color printers you actually enhance the print quality substantially. You don't waste a lot of ink which is absorbed by the paper. The glue on the tape "wets" the ink which causes the colors to look darker, stronger and more vibrant. The overhead film is in this case used as a thick layer of clear varnish which improves the effect even further.

Hiding slot edges

When you build models of for example paper or cardboard, the edges of the material may be more visual than you like it to be, especially if you use slots to create a double curvature effect on a relatively flat surface. The edges look like lines on a surface where there are not supposed to be lines. A simple method to hide these edges is to use an ordinary pencil and drag it along the edge of the paper. The edge will then be darker and less visible. Another method is to mix in some color pigment into the glue. This will also help to cover and hide the edges.

Index

Lundström Design

Ekhagsvägen 7, 104 05 Stockholm, Sweden
Phone: int+ 46 - (0)8 - 15 46 63, 15 47 77
Fax: Int+ 46 - (0)8 - 15 82 85
email: ludesign@algonet.se
http://www.algonet.se/~ludesign