

Model Woodworking Board or Sheet Part

by [woodknot](#) on October 5, 2006

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intro: Model Woodworking Board or Sheet Part

This tutorial covers the creation of a standard woodworking board or sheet goods part. Since most woodworking parts are simple rectangles or linear sticks, an Extrude Boss will be used to create most woodworking parts.

This tutorial is compatible with all versions of Alibre including Xpress.

It is important though to follow certain conventions and standards to ensure consistent part creation and reliable predictable models. Below are some of the most important considerations:

Create Parts and Assemblies with attention to view orientation. The standard Orthographic views of Front, Top, Right and Isometric should reflect those orientations for the Parts and Assemblies. The Front view of a part should be face of a board or molding, and not an edge or end. The Front view of an Assembly should be the front of a piece of furniture, cabinet, or when unclear something reasonable like the face of a bed headboard or footboard. Considering these orientations while creating parts and assemblies will make creation of a drawing easier, as well as provide a smooth and consistent workflow.

Parts and Assemblies should be created symmetrically about the reference planes where possible. There are always exceptions, but you should have a good reason when deviating from this "best practice". This is a best practice that crosses software brand, and is considered a good modeling technique no matter what vendors software you are using. This will allow you to use the "reference geometry" to constrain parts and sub-assemblies in assemblies.

When creating Sheet Goods parts, we prefer to create those parts with the Length oriented horizontally along the X axis, and Width vertically with the Y axis. This convention has its basis in orientation used in Cutlist and sheet goods layout and optimization programs. It is consistent with working with sheet materials on a panel saw, on saw horses or a workbench, or doing layout work. It also helps when dealing with specifying edge banding. By using the convention of grain running horizontally, from Left to Right when there is a grain directionality, assures consistent material handling and will reduce errors. It will also allow design work to proceed with a consistent understanding of grain directionality and orientation from modeling to construction.

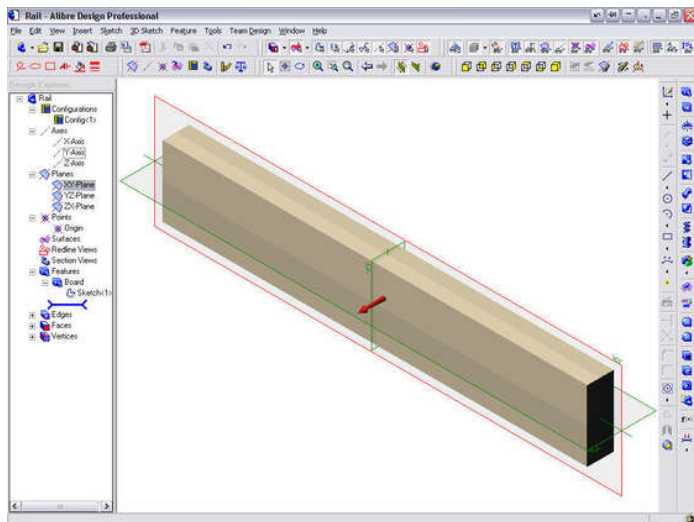
While Alibre does not have material directionality indicators to show grain direction, with this convention you can use the normal arrows from the reference planes to determine face/reverse side and grain direction. Just turn on reference geometry for a part in an assembly and hover over the Planes line item in the Design Explorer to display the normals arrows. We have also begun including a directional arrow on the face of sheet goods parts to aid in properly orientating parts during model assembly. This is a simple arrow shaped line sketch created on the face of the part, and an Extrude Cut 1/64" deep into the face of the part. This indicated Face side as well as grain orientation, and can easily be Suppressed or deleted when modeling is completed.

Lumber can be created with a horizontal or vertical orientation, with a horizontal orientation being common because of sheet good creation conventions. While vertical orientation may be more comfortable when creating legs or stiles, the final orientation of the part in the assembly is determined by assembly constraints to the rest of the model and not the original orientation during part creation. With a little practice, you get used to inserting a part and simply constraining it into position without regard to original orientation, or need to manually rotate the part before applying constraints.

Rather than creating all parts from scratch, we are tending toward using parts saved to a standard "library" parts folder. That way the directional and grain arrow features can be included, as well as a variety of miter features that can be suppressed and unsuppressed as required for baseboards, moldings and other standard uses. We also recommend the "Primitive Wizard" board creation program for Alibre that automates the task of creating rectangular parts (plans.thefrankes.com).

With sketches constructed on the XY plane, a standard board or sheet is Extruded for thickness along the Z axis. While moldings or profiled cross sections are best constructed on the YZ plane and extruded to Length along the X axis. It is a simple matter to orient the display to an isometric view, selected the reference plane on which to sketch, and Activate a 2D sketch on the selected plane. You can then Orient to Sketch plane (if not done automatically) or with a little practice you get used to sketching in an isometric orientation. Remember, construct your parts with consideration to the standard Orthographic drawing views and orientations. It is useful to turn on all the available toolbars.

One last note before creating our board. Woodworking normally only requires the use of a 2D sketch to create even the most complicated of parts. If you are trying to work with a 3D sketch, you should re-evaluate what you are trying to accomplish as it can probably be accomplished in a much easier manner. Besides sketching on the initial reference planes and user created planes, you can use the faces of parts as the plane for a 2D sketch. With Revolve, Sweep, and Loft you can create 3 dimensional shapes using a combination of 2D sketches of profiles and paths.



step 1: Create New Part

Create a new part: FILE > NEW > PART; CTRL + SHIFT + T; or use the New Part icon from Alibre Home window.



step 2: Orient to Isometric

Orienting the Drawing Window to an Isometric View will allow easy selection of Reference Plane or Face on which to create a Sketch.

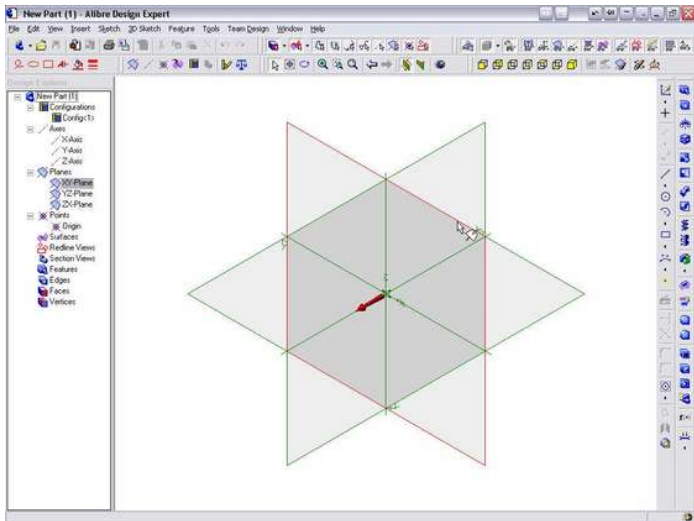


step 3: Select the XY Plane

Move the cursor over the boundaries of the XY plane until the plane figure is highlighted and then Select the XY Plane.

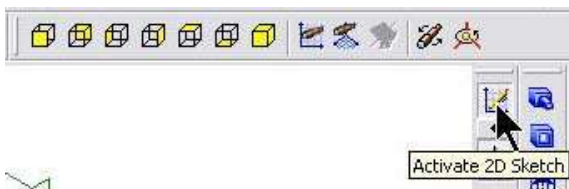
While you do not need to Orient to Isometric to select the XY plane (as it is the default sketch plane), this procedure is handy when selecting other planes and faces on which to construct sketches. This is a useful methodology to understand.

Get used to working with the Isometric Orientation and in a 3D workspace as this will make modeling faster and more efficient.



step 4: Activate 2D Sketch

Select the Activate 2D Sketch icon to activate a sketch on the selected XY Plane.



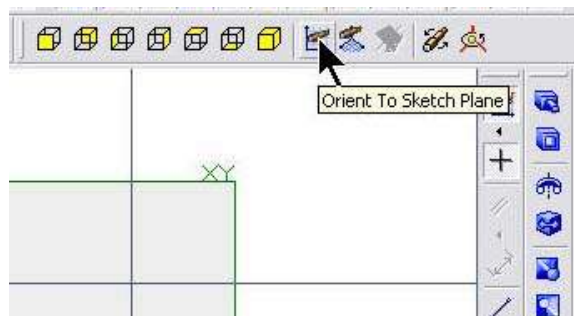
step 5: Orient to Sketch Plane

Select the Orient to Sketch Plane icon; VIEW > ORIENT > TO SKETCH PLANE; or CTRL+SHIFT+Z. This will orient the Drawing area to view the current sketch face on.

HINT: Pressing the Orient to Sketch Plane icon again will reverse the orientation to the backside of the Sketch Plane. One more time will reverse the orientation back to the face side of the Sketch Plane.

If TOOLS > OPTIONS > GENERAL > Design Viewing "Snap to working plane" box is checked, then the display will automatically orient to the sketch plane.

However, it can be useful to retain a viewing orientation when activating a sketch and use the Orient to Sketch Plane instead. With time and practice you can even get used to working while still in the Isometric Orientation.

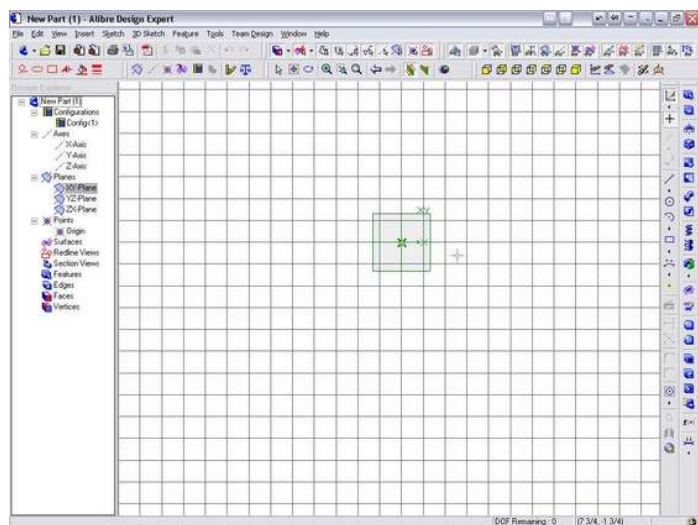


step 6: Zoom Drawing Area to working dimensions

Use the scroll wheel or zoom command to zoom out to where the drawing area is sufficient to contain your part. It is useful to have the Grid Display turned on (Grid Snap OFF) and set to 3" spacing as an aid to judging approximate sizes. If you prefer, you can also just draw, dimension, and press the HOME key to resize the drawing window as you work.

HINT: If you save before creating a solid, you can continue editing the sketch by right clicking on the sketch in the Design Explorer and selecting Edit.

HINT: An entry for your current sketch will not be created in the Design Explorer until the sketch is ended by pressing the Activate 2D sketch button, selecting the Regenerate to Last Feature button, or our favorite the F5 button for Regenerate ALL.



step 7: Draw Rectangle by Two Corners

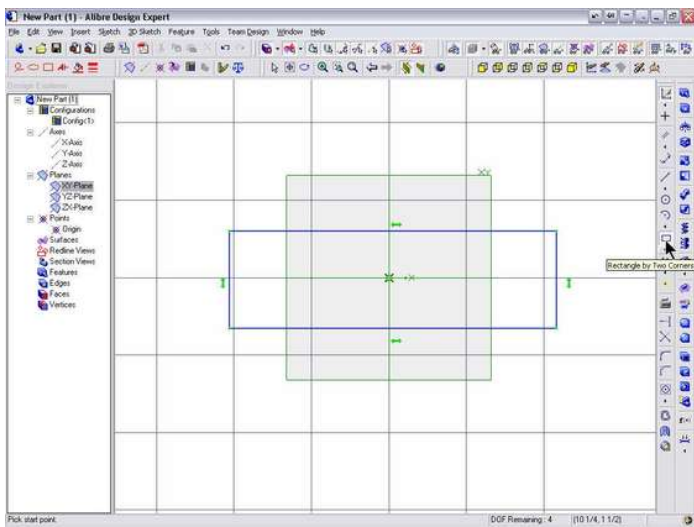
Select the Rectangle Tool, and draw a Rectangle that is roughly centered about the origin. While the size of the sketch does not matter at this point, but we will be creating a part that is 2" vertically by 12" horizontally. Approximate size and proportions can aid during rough layout.

To draw a Rectangle by two corners, select one corner (lower left for example) by pressing and releasing the left mouse button; move to the opposite corner (upper right); and press and release the left mouse button again to specify the second corner of the rectangle.

You can select any two opposite corners, but consistency of technique results in increased speed.

Using the Grid to judge size can be helpful, but snapping to the grid is not recommended as the parts size is controlled by dimensioning the sketch. Un-dimensioned the part can change size even though it was initially snapped to a grid, or coordinates entered via keyboard entry.

Dimensions in parametric modeling specify size rather than simply display a value.



step 8: Constrain Rectangle about X-Axis

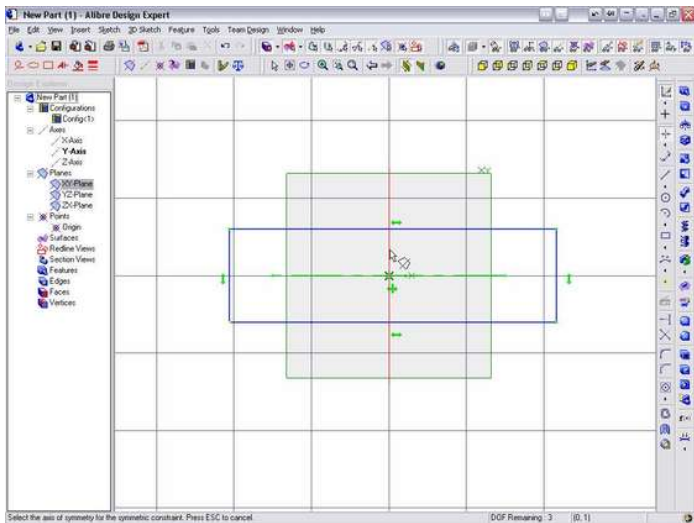
Constrain the Rectangle vertically using the Symmetric constraint about the X axis. Select the X-axis, and then the Upper Left and Lower Left Corner points of the Rectangle.

Modeling symmetrically about the Origin is best practice for any parametric modeling software.

You can select the right corners, but it helps to develop a standard methodology, working Top to Bottom and Left to Right. When it becomes second nature, it will greatly speed up your work as well as improve the consistency and accuracy of a model.

Those users from countries where the written word is other than top to bottom and left of right, may wish to adopt conventions more consistent with their daily lives.

HINT: You could also select the Top and Bottom lines for the symmetric constraint, but that will insert 3 symmetric constraints and symbols rather than one. It will be harder to delete or modify the constraint than using corner points.

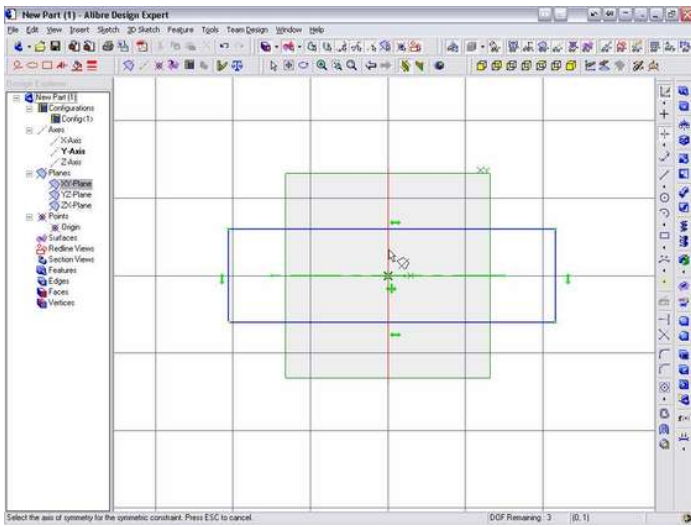


step 9: Constrain Rectangle about Y-Axis

Constrain the Rectangle horizontally using the Symmetric constraint about the Y-axis. Select the Y-axis and the Upper Left and Upper Right corners of the rectangle.

You can select the Axis from the Design Explorer on the Left as well as from the Drawing Work area.

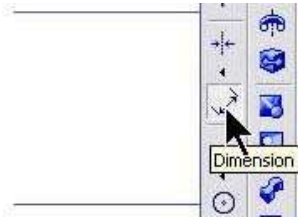
HINT: It is prudent to fully Constrain a sketch before dimensioning, as that will prevent the sketch from shifting with the addition of the dimensions, and in some cases prevent the sketch from distorting itself while being dimensioned.



step 10: Select Dimension tool to dimension Width of Board

To Dimension the Width of the Board, select the Dimension tool, and dimensioning the Left Vertical side of the Rectangle.

Select the Dimension tool.

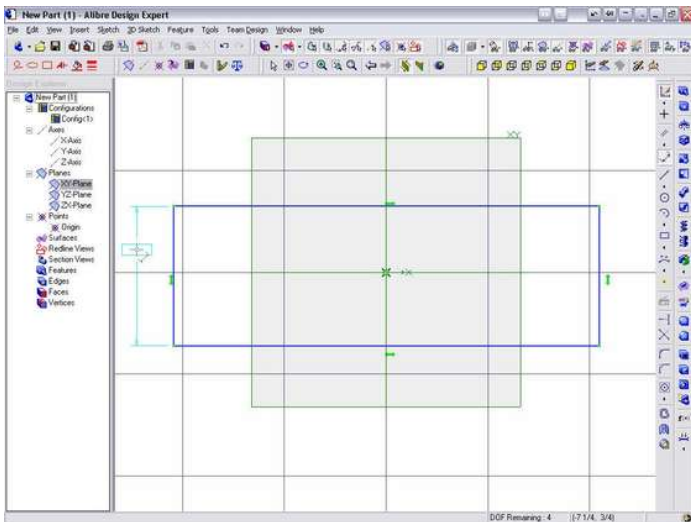


step 11: Dimension Vertical Side of Rectangle

Hover over the Left (or Right) side Vertical line for the Rectangle until the line highlights.

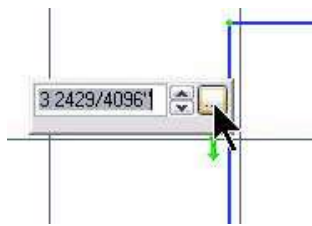
Pick the Left (or Right) side Vertical line of the Rectangle.

Move off the line and Pick another point to locate the dimension.



step 12: Specify the Vertical Dimension with the Equation Editor

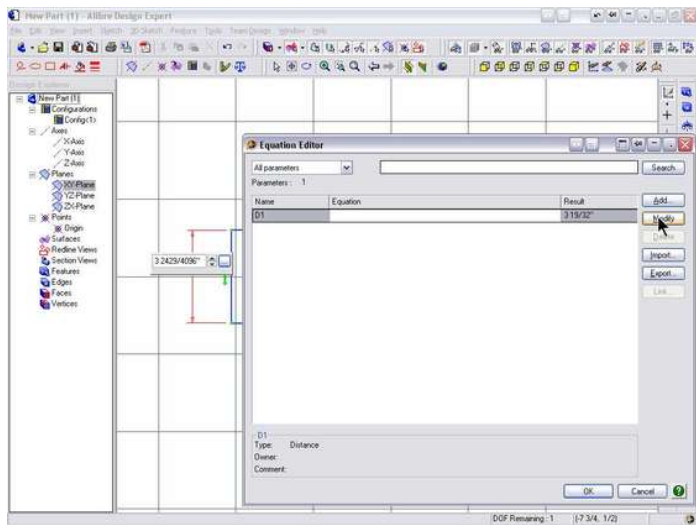
Select the ... box to the right of the dimension value to enter the Equation Editor. This will allow specifying the Parameter name as well as value for this dimension.



step 13: Use Modify Button to make changes in Equation Editor

Select the Modify button to make changes to the current dimension parameter.

Use the Modify dialog rather than just making changes to the parameter name and equation value because there is additional error checking and data validation. If you enter an invalid parameter name or value directly you may not notice the parameter name has reverted to the original name. With the Modify dialog, the OK button will not be available until a valid name is entered. Valid names start with a letter, and do not contain spaces, with common convention being the use of an underline, or simply capitalizing the first letter of a new word.



step 14: Enter Width for Parameter Name & 2 for Equation Value

Change the Name of the Parameter to Width. Using a standard parameter name helps eliminate confusion as to the purpose of the parameter, as well as allows a parameter to be included in the Bill of Material (BOM) automatically created from the model.

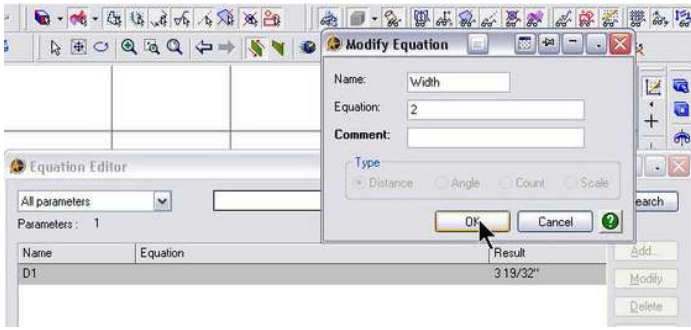
Tab and enter a value of 2 for the Equation field of our 2" wide tutorial Rail part. Do not enter the " sign for inches as you should use the two letters 'in' instead. You can enter other units directly, such as 50mm for a millimeter based part. The default units defined in the properties will be used when units are not specified with the entry.

Tab off the Equation value entry field to complete the entry, and select the OK button to accept the modifications. The TAB key signifies to Alibre that a field entry is complete, and in some cases is required in order for the OK, APPLY, or other buttons or features to become active in a dialog box. Additionally, it is a good idea to end an entry with the TAB key as it will help avoid encountering certain problems where data validation may be lacking.

Select the OK button to close the Equation Editor window and return to the dimension entry dialog.

HINT: It is important that the current parameter is highlighted when exiting the Equation Editor dialog window! If any other parameter is highlighted, the value of that parameter will be replaced as the value of the current parameter. This is a feature that allows a value of a parameter to be specified by selecting another parameter from the Equation Editor dialog window.

Enter the Equation Editor via the icon or Ctrl+E when making changes to parameters, and avoid making changes to other parameters when using the Equation Editor to specify the value of a particular parameter during dimensioning.



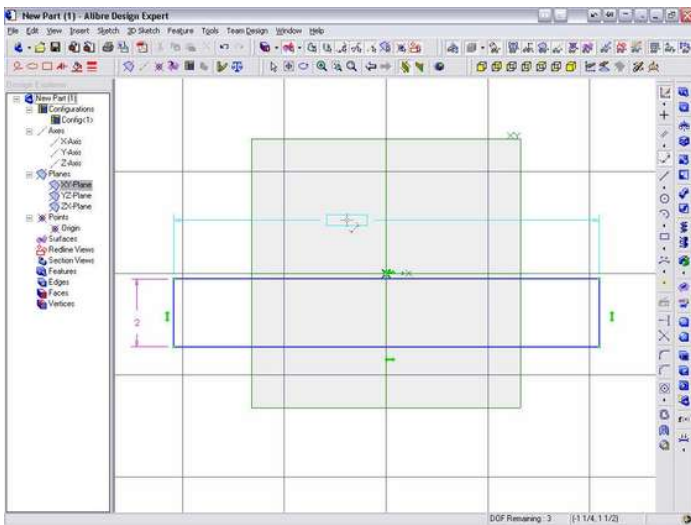
step 15: Dimension Horizontal side of Rectangle

Dimension the Length of the Board by dimensioning a Horizontal line of the Rectangle.

Select the dimension tool.

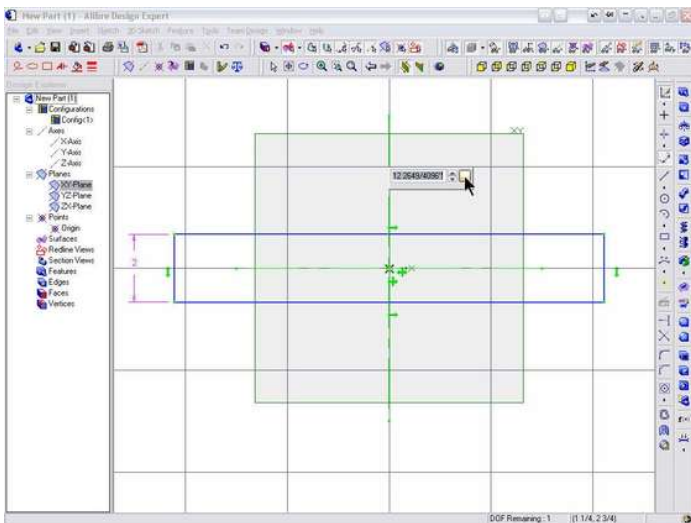
Hover over the Top (or Bottom) line of the Rectangle, and Pick the line.

Pick a location in the drawing area for the dimension.



step 16: Specify Length Parameter using Equation Editor

Select the ... to the left of the dimension value to specify the Parameter name and Equation value using the Equation Editor.



step 17: Use Modify Button to make changes in Equation Editor

Select the Modify button (or Alt+M) to modify the current parameter.

Change the Parameter Name to Length.

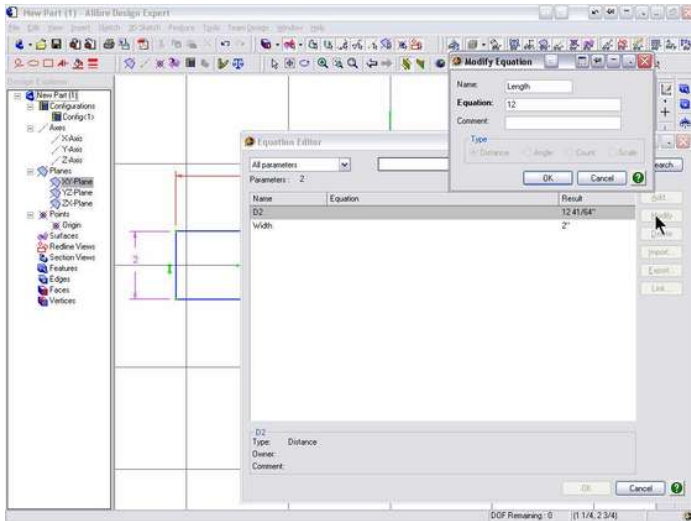
Change the Parameter Equation value to 12, for a 12" long part.

Select OK to close the Modify dialog box and accept the changes.

Select OK to close the Equation Editor dialog box and return to the dimension box.

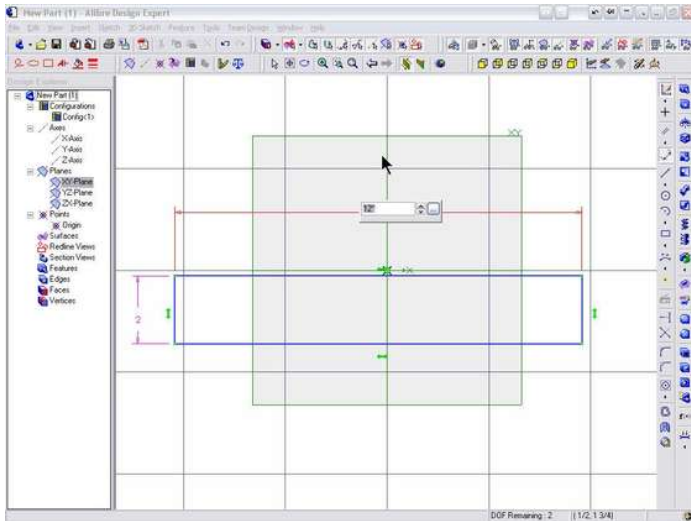
HINT: It is important that the current parameter is highlighted when exiting the Equation Editor dialog window! If any other parameter is highlighted, the value of that parameter will be replaced as the value of the current parameter. This is a feature that allows a value of a parameter to be specified by selecting another parameter from the Equation Editor dialog window.

Enter the Equation Editor via the icon or Ctrl+E when making changes to parameters, and avoid making changes to other parameters when using the Equation Editor to specify the value of a particular parameter during dimensioning.



step 18: Close the Dimension box

Press Enter to close the dimension box and accept the dimension.



step 19: Extrude Boss to create Solid Feature

Select Extrude Boss icon to create a solid feature for our Rail part.

We prefer to have "Reorient on Extrude" unchecked under Options in order to retain any current view orientation that we have set. Often choosing an Isometric orientation to verify the extrude is being created as expected.



step 20: Extrude Boss Dialog

The sketch selected in the Sketch box should be the current sketch containing the rectangle. If the sketch is not currently active from being created or edited, pick in the sketch dialog box and then select the sketch in the Design Explorer.

In the Extrude Boss Window under Type:, use the drop down to select MidPlane as the type of extrusion. This is keeping with best practice of constructing symmetrically about the reference planes.

In the Depth section, Select the ... box after the Depth parameter to modify the parameter using the Equation Editor.

In the Equation Editor, Select the Modify button (or Alt+M) to modify the current parameter.

Change the parameter name to Thick. This is a common shortening of Thickness for easier naming relating to the thickness of parts

Change the parameter Equation value to 3/4, for a 3/4" thick part.

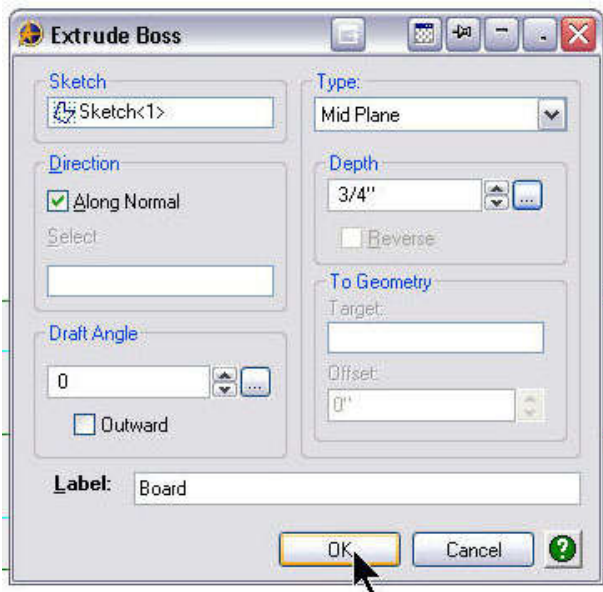
Select OK to close the Modify dialog box and accept the changes.

Select OK to close the Equation Editor dialog box and return to the Extrude Boss dialog.

Specify the Label of the part as Board or Part, and select the OK button.

Avoid specific names like Rail for the feature, as you will often re-use parts and most likely save this same part as Stile with only a few changes of parameter values.

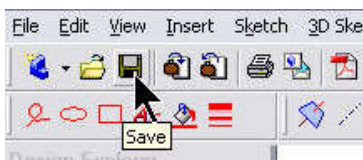
The file name for the part is what is displayed in the Design Explorer as well as the Bill of Material, and not the name of the particular feature that creates the board. It can be disconcerting to users who open a part named Stile and find the main feature named Rail.



step 21: Save the Rail Part

Save the Part file under the name of Rail.

Verify the location or folder for the Save before completing the Save. Alibre defaults to the last folder used for a save. You can create a new project folder using the New Folder button.

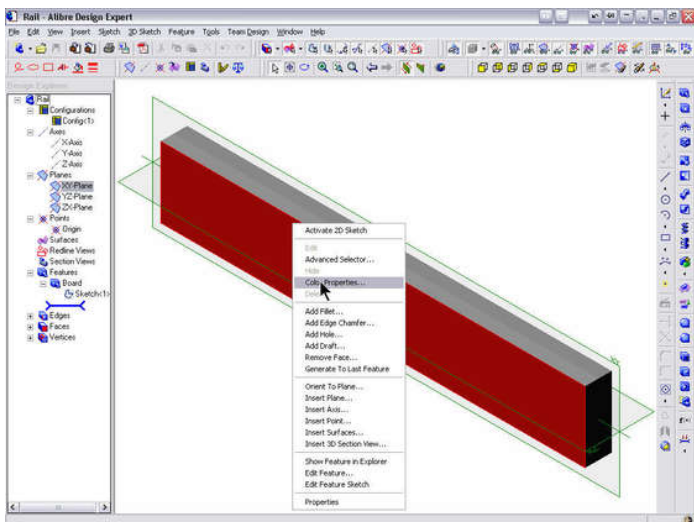


step 22: Part Color Selection

To complete the part, assignment of a Color will enhance the usefulness of a part. Although a piece may be constructed of Walnut, the darker colors tend to obscure model details and use of lighter colors simulating maple, cherry, and oak are a better choice.

Right Click in the drawing area over the part, or on the part in the Design Explorer, and select Color Properties from the pop-up menu.

Varying the color slightly between parts also aids in differentiating parts from each other, while not detracting from the overall look of the model. We use a 3 level color variation of standard, lighter, and darker. In general, the following color variation scheme has worked well for our models. The horizontal parts using the standard color, the vertical parts using the darker color, and parts like the back and door panels using the lighter variation of a color. Adjustments can then be made based on the model construction where additional color variation is desired. Color Properties can also be specified in the Assembly, which will over ride the parts color without actually changing the color of the original part. This is useful when a single part is used a multiple of times and a color variation is desired to highlight the construction.



step 23: Enter specific Color Values

Select the Color button from the Color Properties dialog window and the Color specification window will open in order to specify a color.

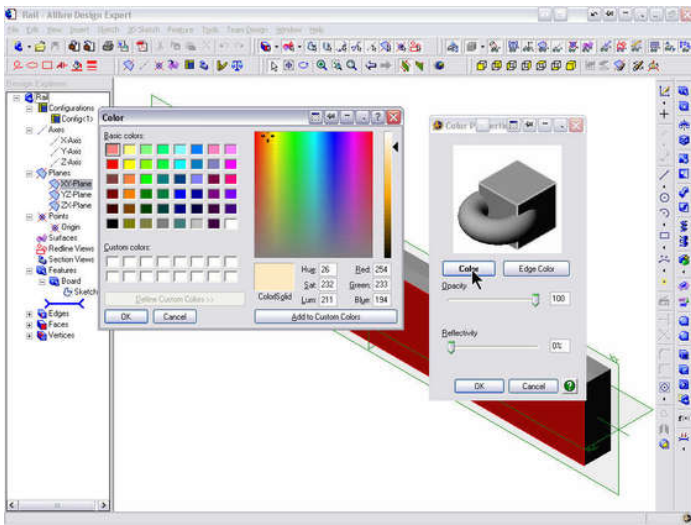
Select a standard color, or enter a custom color from the custom color dialog. Below are Red; Green; Blue values we use for representative wood colors.

You can save a specific color by first selecting a color box, entering the color values, and selecting Add to Custom Colors button.

Not selecting a box before hand could overwrite and existing color definition, and selecting a color box will replace the current color values with those defined for that box. You need to select an available Custom Colors: box, enter the desired values, and then select the Add to Custom Colors button to specify and retain custom color definitions.

Colors definitions used by Kelsey WoodWorks to represent wood species are:

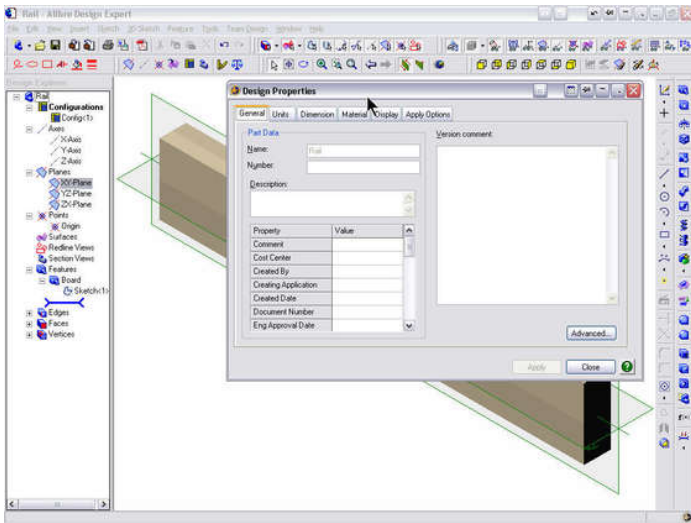
Wood	Red	Green	Blue	Red	Green	Blue	Red	Green	Blue
Maple	254	233	194	254	239	211	254	227	177
Cherry	216	141	112	201	159	126	185	138	104
Oak / MDF	230	189	104	235	201	131	225	175	72
Walnut	122	64	36	122	64	36	122	58	30
Mahogany	244	148	87	238	139	74	244	148	74
Padauk	203	53	16						
Wenge	137	68	52						



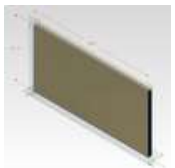
step 24: Part Data Properties

The final item for a complete part is the definition of Part Data Properties. These properties will be included in the Bill of Material (BOM) and can be exported to other programs such as Cutlist software to provide complete information for a part.

Part Data fields are available under FILE > PROPERTIES and the GENERAL tab, or the key combination of ALT+ENTER and the General tab. Standard field names are defined to assist in organizing the data, but any field can be used to store data appropriate for your needs or software application. An example being the use of the Stock Size field to store the Stock definition data for CutList Plus. While we originally "mapped" our data to fields with similar names, we later changed our mapping to fields that were located closer together to make data entry easier. Mappings of data fields for various cutlist programs are available for download at our website: www.kelseywoodworks.com.



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Comments

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woodworks says:

Oct 6, 2006. 6:23 PM [REPLY](#)

I really like working with Alibre because it is like working in wood. The solids I create can be worked with techniques like woodworking, and I can build a virtual prototype before even cutting my first piece of wood.

This has saved me a lot of trial and error, as well as time doing mockups to verify the proportions and correctness of a design. So much better than 2D CAD because I can rotate the design and look at it from all sides.

It all becomes so much easier after the first time or two, so stick with it as it is worth it.



woodknot says:

Oct 6, 2006. 5:54 PM [REPLY](#)

Sorry for the length of the tutorial, but we tried to squeeze in all the fundamental knowledge of nearly two years of doing woodworking with Alibre, and a lifetime of experience with specialty woodworking and CAD software. There have been a lot of woodworkers asking us how to best use Alibre for Woodworking, and we wanted to save other users from learning all this by trial and error as we had to.

There is more information than can probably be absorbed in one pass through the tutorial, but it was intended to address all the aspects of using a parametric modeling tool for woodworking. Start off with the other tutorials for basic techniques, and come back to this tutorial after you have gained familiarity with the basics of the software. A good place to start is the QuickStart Guide video available at the Alibre Xpress website and the other Instructables at this site.

Woodworking can be done with a simple rectangle sketch and Extrude to create a board. But, if you want to model an entire kitchen cabinet, furniture project, or entertainment center, there are a lot of other considerations to address to take the model from the rectangular parts to a completed cutlist and sheet goods layout. This tutorial is more of an application of Alibre to woodworking, rather than an introductory course on how to use Alibre.

While Sketch Up is a very good tool, we prefer Alibre because of the ability to drive models with formulas and associations between parts. We moved to Alibre because it was so difficult to make changes to an initial design with other software. Now we can change the dimensions of our designs with a simple change of a value or two. An entire cabinet will automatically adjust the dimensions of all the pieces when we only change the cabinets width or other overall dimension.

Also, visit our website and download the free video tutorial on how to model a simple keepsake box project. Sit back and watch how easy it is to model woodworking after you learn the basic techniques. Visit www.kelseywoodworks.com for more information on Alibre as applied to woodworking. Besides it is a lot of fun too!



cad_crew says:

Oct 6, 2006. 3:00 PM [REPLY](#)

Bear in mind, that while this is a fairly in-depth tutorial for a relatively simple object, it is really more about learning the foundation of parametric solid modeling.

All dimensions in parametric modeling are variables (i.e. parameters) that can be changed once the initial design is completed. This enables very rapid changes and what-if scenarios. This also introduces the concept of associativity, where when you create an assembly, or a 2D shop drawing from the 3D model (which is supported in the free Alibre Design Xpress product), then everything updates automatically when one element is changed.

SketchUp has a nice user interface and a lot of cool features, but it has no notion of parametrics or associativity. Don't take it as a negative comment about SketchUp, they are just two different classes of products.

Indeed there is a learning curve with 3D parametric modeling, although you may find it is not as tough as it might seem at first glance. Once you learn the basics of parametric modeling you can quickly model up a 3D design and then just tweak the dimensions to change it and then automatically generate variations on a design, or families of parts, or a cut list for fabrication.

Parametric modeling may not be your cup of tea, but if you are interested in learning a skill and a product that you will find very productive if you do any sort of design or fabrication with any frequency.

Hope this helps.



gigman says:

Oct 6, 2006. 8:27 AM [REPLY](#)

I agree with radiorental, SketchUp is more suited to do drawings like this.



westfw says:

Oct 6, 2006. 1:36 AM [REPLY](#)

Wow. On the one hand, I've been interested in documenting some woodworking projects using a 3d cad package (preferably one of the CAD packages with a freeware or low cost license. I tried gizmolabs DesignIntuition, which I like but found too expensive for "playing"), And this instructable is a nice step in that direction. But the length and complexity is daunting! I don't want the drawing to be harder than the bookcase was in the first place! (although it was that way before CAD too, I guess...)



radiorental says:

Oct 6, 2006. 6:32 AM [REPLY](#)

for that sort of documenting, google sketchup is the ticket. Half way between drawing and CAD
