

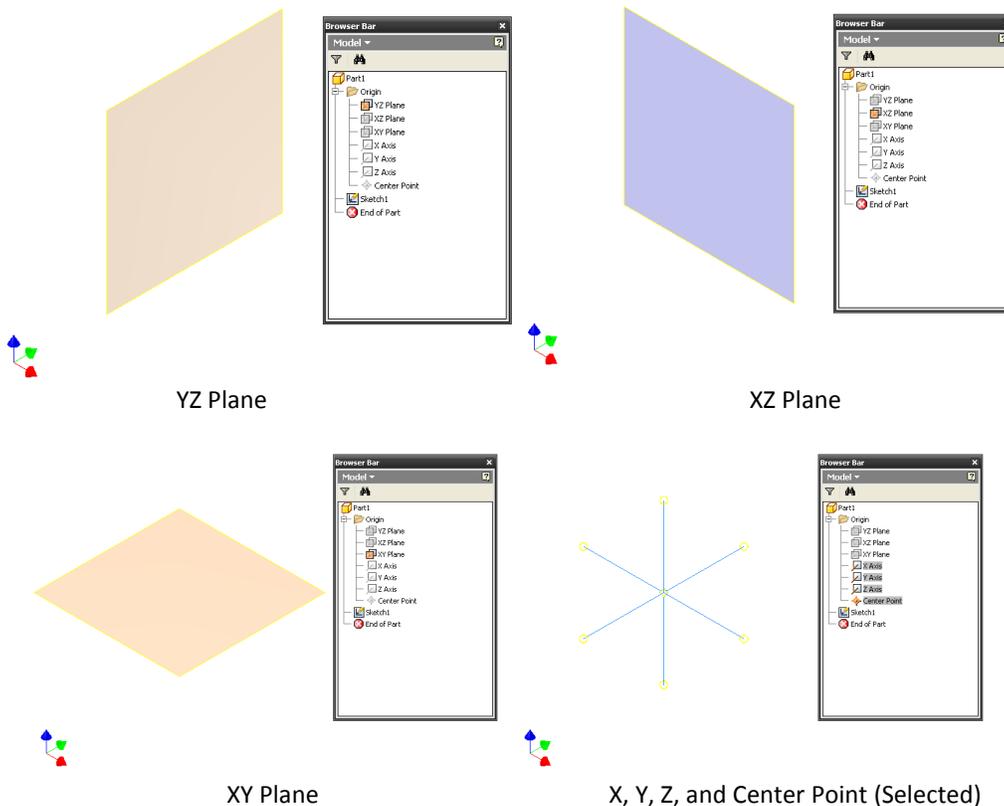
The BORN Modeling Technique

There are many ways new users approach the Inventor software. Some users come from a background where anywhere in space to model features is the standard. Unbeknownst to them it does not necessarily make their models more stable than the method I am about to describe.

The BORN technique or (**B**ase **O**rphan **R**eference **N**ode) is the basic concept of creating your features around your Origin References; Namely your XY, YZ, XZ planes, XYZ axes, and Center Point (0, 0, 0) of your model. These features are always the first in a new part or assembly file and help you get your bearings in Cartesian space. These reference datum planes can be used for dimensions as well as geometric constraints in the software.

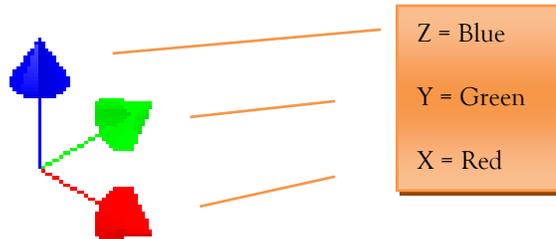
Origin Features

Below are some images of the Origin features. This was accomplished by simply starting a new part file and finishing the first sketch without creating any geometry. I turned on each plane individually by expanding the Origin Folder in the Browser Bar and RMB (Right Mouse Button) on the plane I wish to show and selecting the Visibility toggle.



Where Am I?

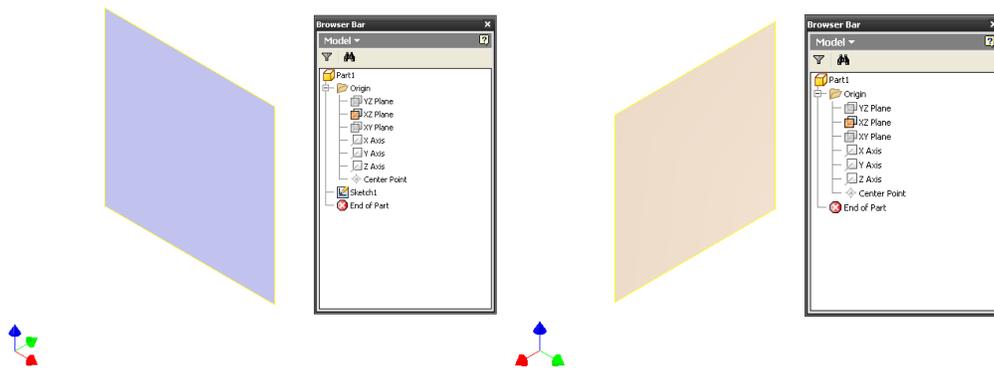
One of the biggest struggles I find with new users is trying to find the correct orientation of their current view or their model while working in it. For them I recommend this trick with the Origin Triad:



Make a mental note: XYZ = RGB

Normal of the Planes

This next idea can be said for any Work Plane, not just the Origin planes. When viewing a plane there are actually two different colors associated with it. The color you see depends on which side of the plane you are viewing it. From the Normal (positive direction) side of the plane you will see one color and from the bottom side of the Plane you will see another. The Normal is defined by its orientation of creation or in the case of the Origin planes its relation in Cartesian space. Notice the XZ Plane below. We are looking at it from the bottom side of the plane so in this case we see a different color than the other planes. If we rotate the model around a little we can see that it is a different color on the opposite side.

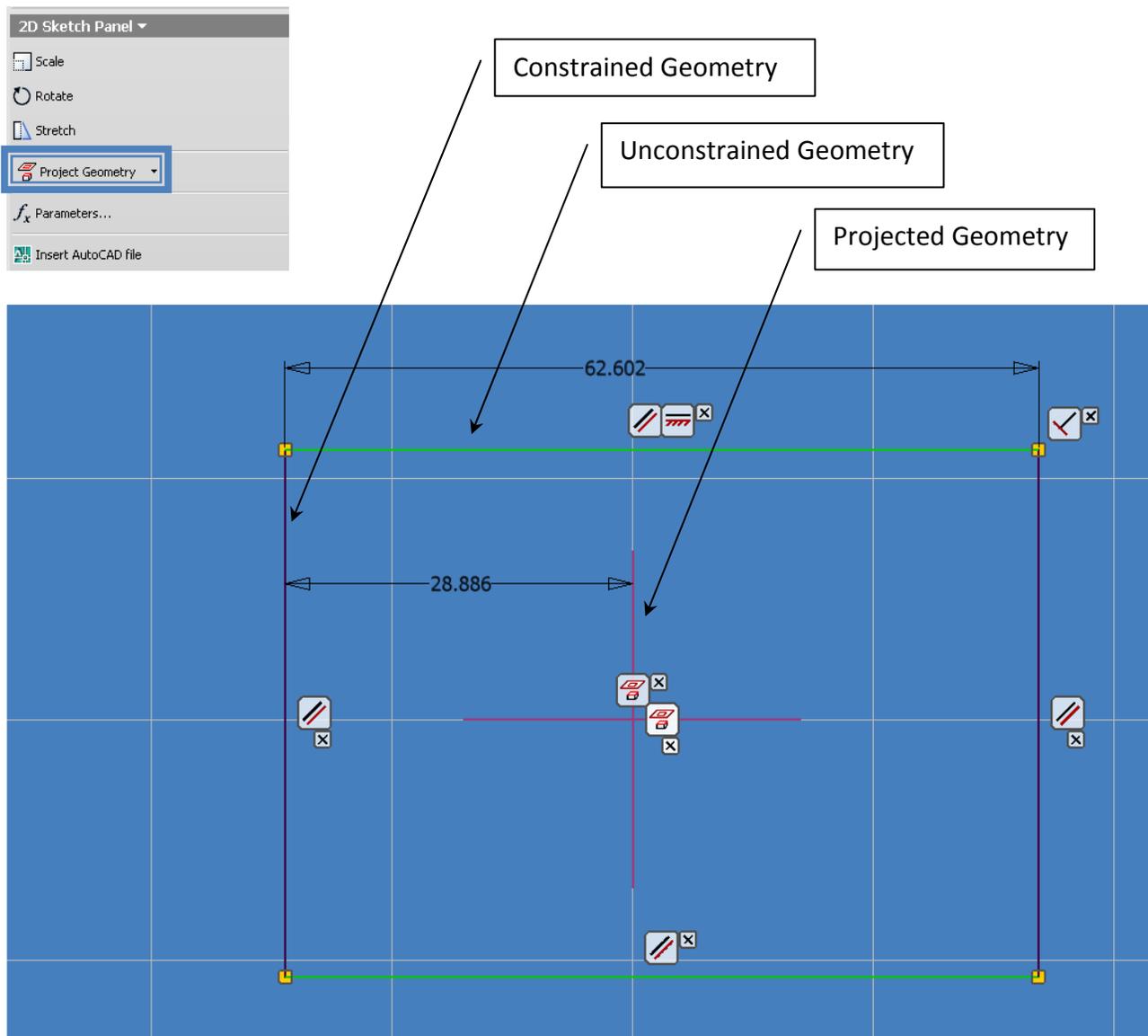


Size of the Origin Planes

The size of your Origin features may seem like your Part file is extra small. Inventor actually Auto-Resizes the planes based on your model as you are creating it. This is another toggle when you RMB on a plane in your Origin folder and is defaulted to be on with a default template. Origin planes are in fact infinite in direction which follows the definition of Cartesian coordinates.

Using Origin Geometry

Visualizing the Origin geometry is one thing. But simply having it turned on with the Visibility toggle is not enough for us to use it. In order to get the geometry for sketch references for dimension or geometric constraints we have to use the Project Geometry tool located in the 2D Sketch Panel. This command allows us to project the geometry as a reference object in our sketch. Origin features can be selected visually in the Graphics Window if they are turned on or by selecting them in the Browser Bar if they are turned off. Either way they will be selected. The projected lines will appear in your sketch as a different color than your constrained and unconstrained lines. I changed my Color scheme (Tools → Application Options → Colors tab) to an easier to see coloring for the lines. Depending on what your color scheme is the lines will have different colors. I have also turned on my geometric constraints (F8).



Taking the Next Step

When should you use the Planes and Axes and when should you just use the Center Point? When the Planes or Axes are chosen they essentially are projected so that they are shown in the sketch where they would intersect the Sketch Plane. This is very helpful for setting up Geometric constraints such as Collinear and Symmetry as well as General Dimensioning to the Origin features. If just the Center Point is used it changes the type of constraints that can be used (Symmetry is no longer available but Horizontal and Vertical alignment as well as Coincident can be used). Different design intents mean one method or both will be chosen.

The Base Feature Using BORN

After a strong fully constrained initial sketch is created, the next step is to create a Base feature to start building your model and subsequent secondary features. You may have wondered why sometimes the direction of the Extents of a Base feature need flipped in order to work the way you wish. This is all due to the Cartesian orientation of your sketch and your current viewing of the model.



Creation Directions From Left to Right (Direction in the Normal, Direction away from Normal, and Direction splitting the difference between the Normal and away from it)

Saving Time

The process of starting a new part and then telling the software each time to project the Center Point or the Origin planes can become rather repetitive. In order to avoid this mundane task each time you start a new file there are some shortcuts. One approach is to change your template part files to have the Origin geometry projected and/or Origin features Visibility turned on when you start a new file. So every time you create a new part file you will have your origin features automatically projected and available for use in your sketch. Another method is to use the Application Options. There is a setting under the Sketch tab here that will allow an “Autoproject origin on sketch creation”. This will project the Center Point to every sketch you create.

Final Remarks

The BORN method is a really nice approach to modeling because it puts a strong sense of orientation and structure to your model rather than creating a “floating” base feature. It is the best way to create a fully constrained sketch in order to create stable 3D parametric models. This method also aids in the Assembly process if you like constraining your Origin features together or if you plan on using some of the more advanced workflows in Inventor including Derived Parts and Skeletal Modeling.



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Mark is an Application Engineer with IMAGINiT Technologies, specializing in manufacturing environments. He has implemented the Autodesk® Manufacturing products with several industries including the blow/injection molding, automotive, and custom machinery markets. Inventor has been a profound augmentation in his abilities allowing him to bring 3D digital prototyping to the forefront of the industries with which he has interacted. He has extensive experience and a comprehensive understanding of the technical, practical business and human dimensions of implementation. He is an effective and skillful communicator, consulting with his clients to help achieve their business objectives. Mark is an ATC certified instructor and he provides training, support, and implementation. Mark is PSE and ATC certified in AutoCAD®, AutoCAD® Mechanical, AutoCAD® Electrical, Autodesk® Data Management, and Autodesk® Inventor®.



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