

Best Practices for Sketching in Alibre Design

Good rules to follow during the sketching phase of the solid model creation process.

1 When Sketching Geometry for Solid Parts;

1. Create the profile as close to the final constrained and dimensioned part as possible.
2. Add geometric constraints (tangency, perpendicularity, etc.), or turn on the automatic constraints functionality.
3. Add dimensional constraints (diameters, then radii's, then angles, then lengths). See figures 12 thru 14 for hints on the correct way to dimension sketches.
4. Analyze the sketch. This is always a good idea, even in the case of simple sketches.
5. Dress-up features. Do not create fillets, chamfers, or drafts in the sketch; instead create them in the solid model. The only exception to this guideline under normal circumstances is when such elements are integral parts of the general model shape.
6. Sketch simple and coherent profiles (maximum of 15 elements). Although more complex sketches can be created, a series of simpler sketches will make future part modification easier.
7. Only one profile per sketch.
8. One part function per sketch.
9. Try to avoid dimensional links to solid models, instead use reference elements (axis, lines, planes, etc.) whenever possible.
10. Do not use the origin point as a sketch entity point, sketch next to it and use a coincidence constraint.
11. Modify sketch elements instead of replacing them.

Explanations

1.1 Create a profile as close to the final results as possible (shape and scale).

When sketching a profile, try to create a shape that is as close to the desired shape as possible. This will make it easier to add dimensions and constraints when arriving at the final shape. Setting the grid in correct relation to your part's size, will make sketching more efficient.

1.2 Adding Geometric Constraints

Give some thought to exactly how the different parts of your sketch relate to one another before applying geometric constraints. It is often possible to reduce the number of constraints while still maintaining tight control over the shape of the sketch

If you've turned on the Automatic Constraints feature, be cautious in adding other geometric constraints as this often leads to an over-constrained sketch.

1.3 Adding Dimensional Constraints

Good dimensioning practice recommends creating ‘distance’ constraints in lieu of ‘length’ constraints. Dimensioning a single elements length (see figure 12) only constrains the length of the element, and not the position of the part in the sketch. Instead, select the support or limiting elements to fully constrain the sketch element.

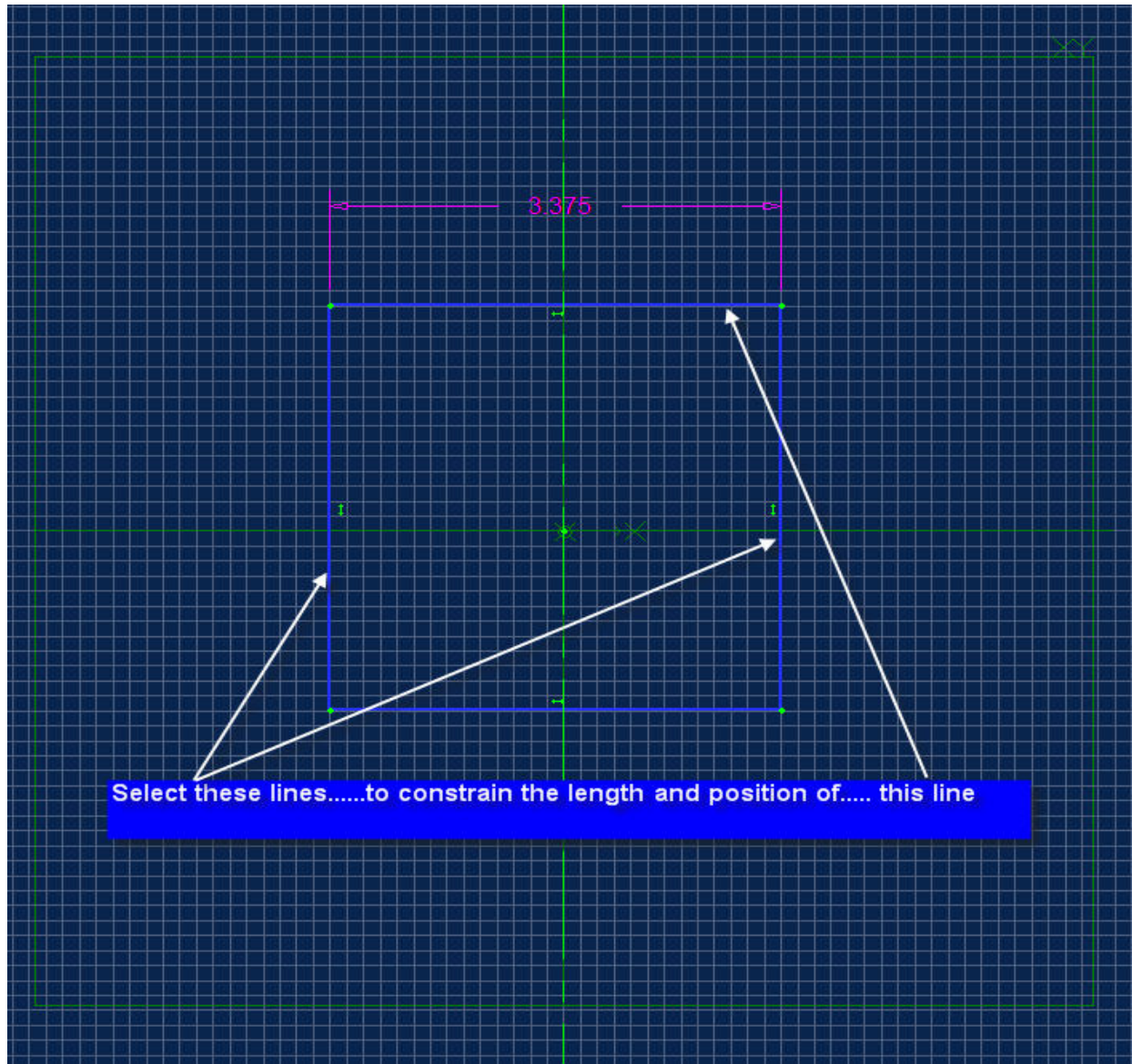


Figure 12

The order of adding dimensional constraints should be as follows;

- Radii (diameters)
- Angles
- Lengths (distances)

It is also highly recommended that you fully constrain the sketch. The method used to verify that a sketch is fully constrained is to reduce the Degrees of Freedom (DOF) of the sketch to 0. As you add constraints to a sketch, the degrees of freedom the DOF will decrease. (See

figures 13 and 14). When constraining a sketch it is necessary not only to dimensionally constrain the various elements, but also to also dimensionally constrain the sketch to the geometric support. (axis or construction planes)

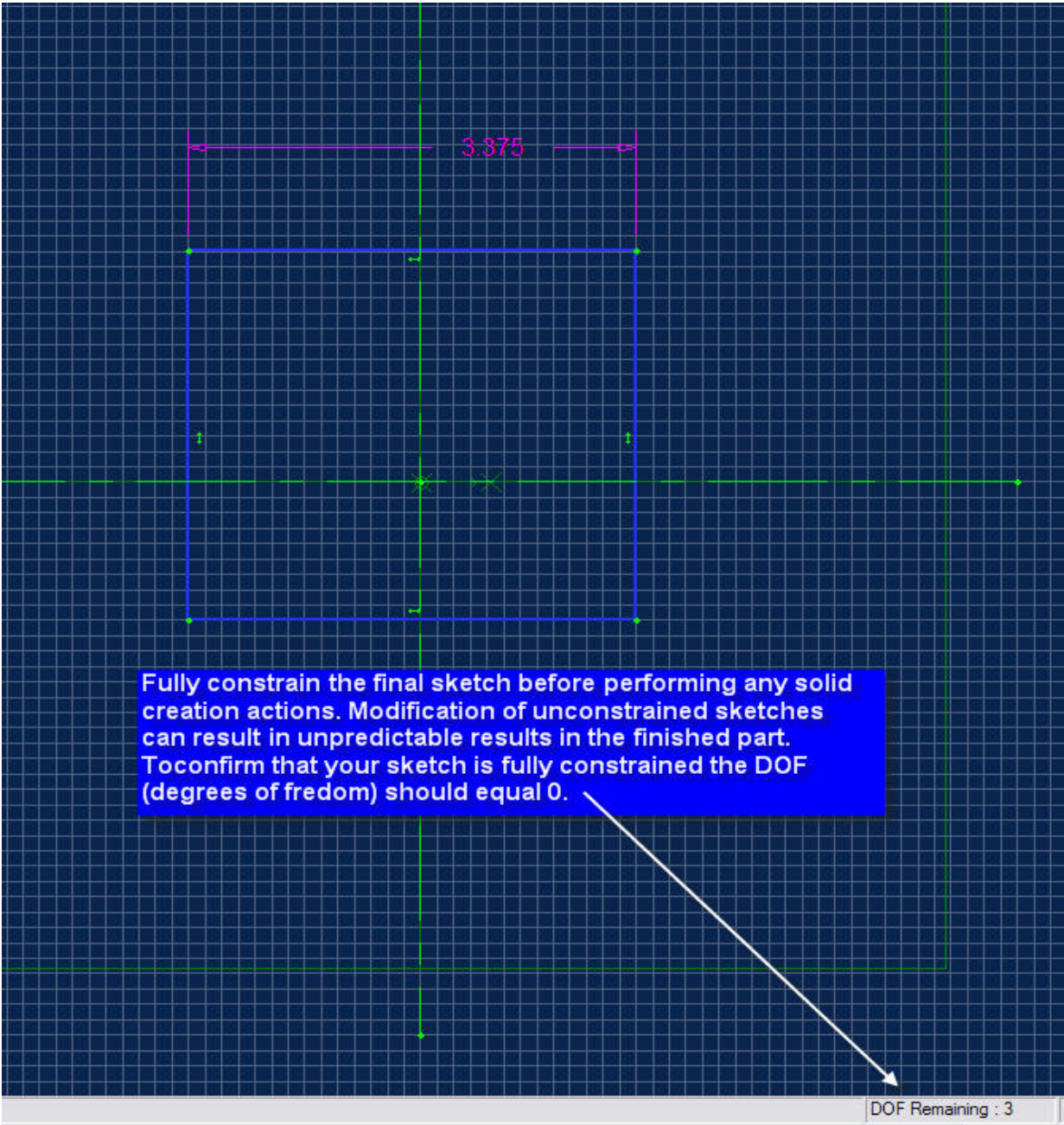


Figure 13

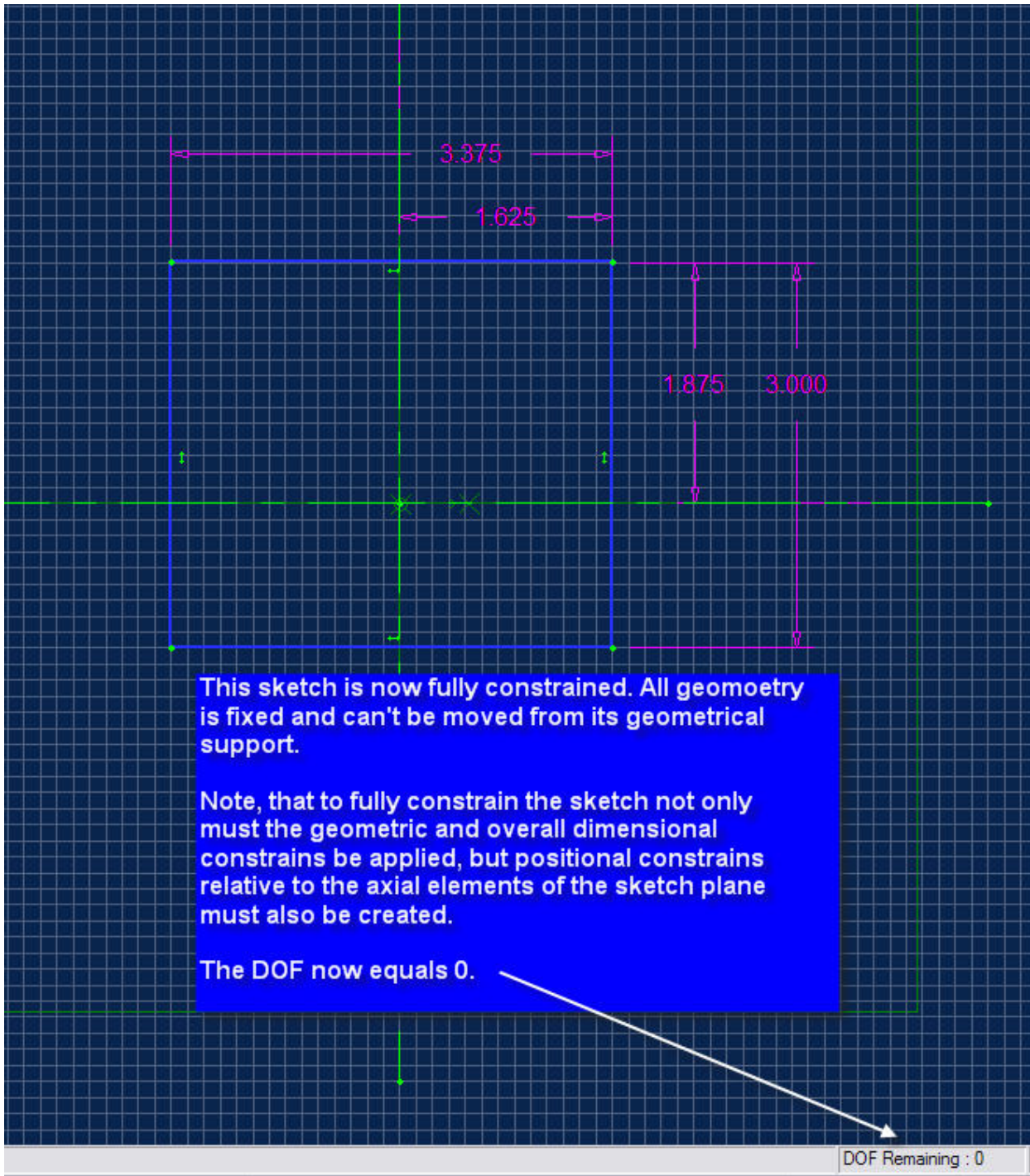


Figure 14

1.4. Analyze the Sketch

After the sketch is fully constrained and before you initiate, any of the solid creation commands, it is a good idea to analyze your sketch. You'll pick up any errors, i.e. open loops, crossing lines, etc. before they pop up as an error. (Figure 15) Sketching errors encountered will prevent you from creating a solid model, so early detection and correction of sketching problems is a way to improve your efficiency.

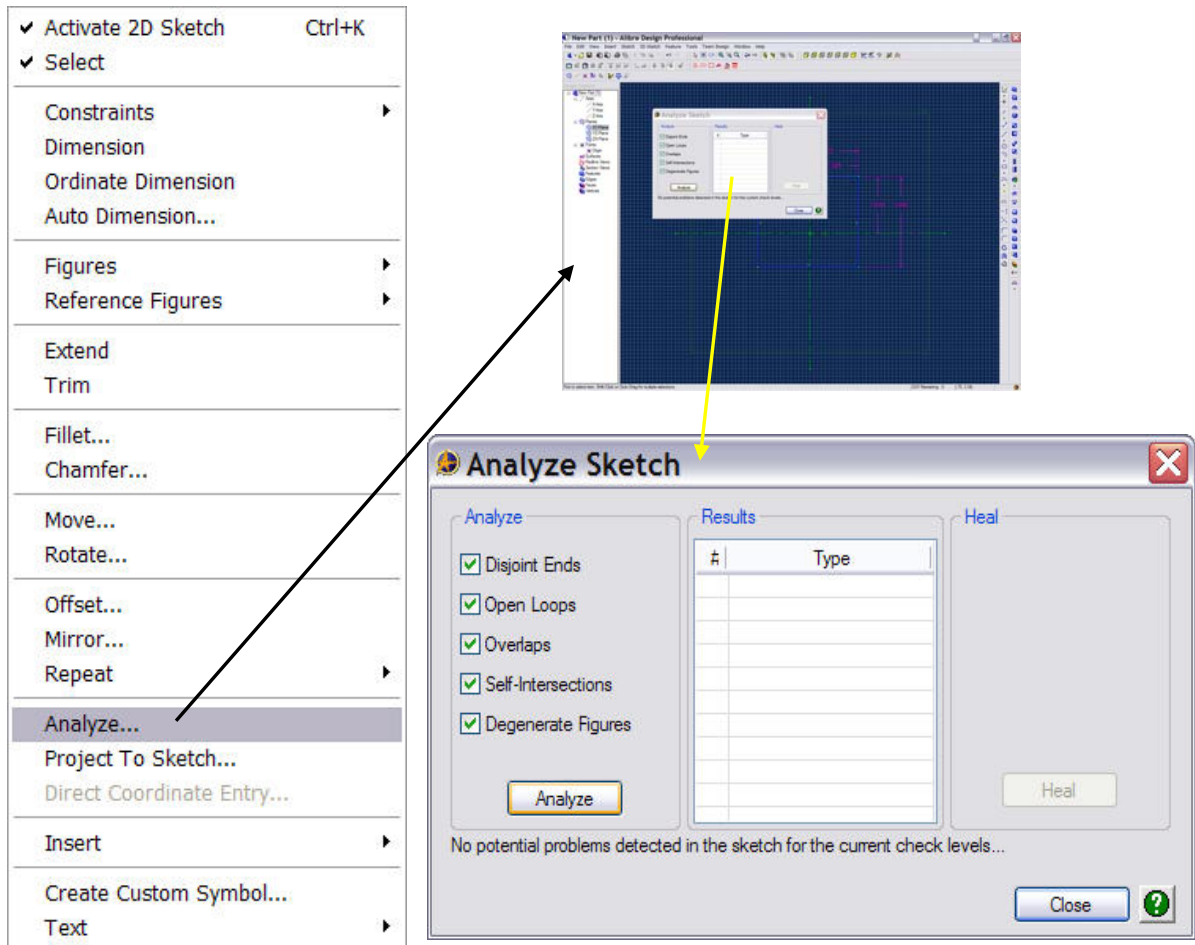


Figure 15

The 'Sketch Analysis' feature provides 'Healing' capabilities but they are limited to the type of error(s) involved.

1.5 Dress-Up Features

Unless a chamfer, fillet or draft angle is an integral aspect of the part design; these features should always be added in the part and not in the sketch. These features should be created as late in the design process as possible to make editing easier and to reduce the impact of modifications on the part. (See figure 16)

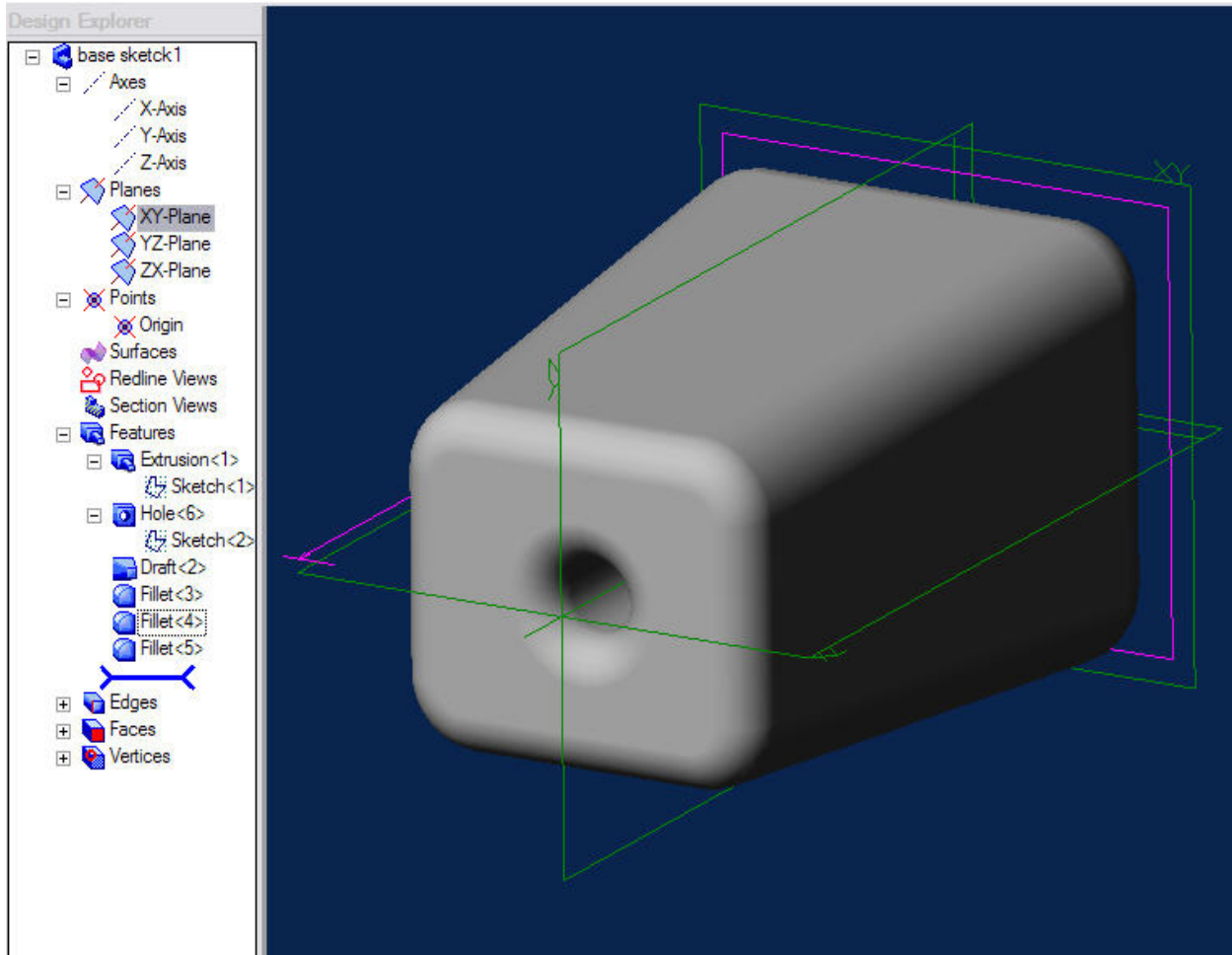


Figure 16

1.6 Simple, Coherent Sketches

In place of complex sketches with a large number of elements and inter-related constraints, always strive to create simple sketches with a minimal number of constraints. This makes future editing of the sketch easier and reduces the number of un-predictable part modifications that often occur with complex sketches. It is also good practice to use the sketch functions such as pattern, and mirror and the equal (=) constraint to simplify sketching.

1.7 One Profile per Sketch

The use of one profile per sketch ties in with the practice of using simple sketches to represent part features. Although it is possible to create sketches using more than one profile, future modification of the part may be more difficult due to confusion related to sketch-part fun

1.8 One Part Function per Sketch

Again, the less complex your sketches and inter-related part functions are, the simpler future modification will be. Remember, you will more than likely **'not'** be the person making the modifications and it will be much more efficient if the part design follows a logical creation methodology.

1.9 Reference Elements

Creating reference elements, planes, axes, or points, for use in sketching is a better choice than using solid edges, faces, or vertices. If you use a face on a solid part for a sketch support, you must be sure that the feature you create will evolve in the same way as the face you've selected.

1.10 The Origin Point

Refrain from using the sketcher origin point as a primary sketching point. Instead, sketch near it and use a coincidence constraint to anchor your geometry to it. (See figure 17)

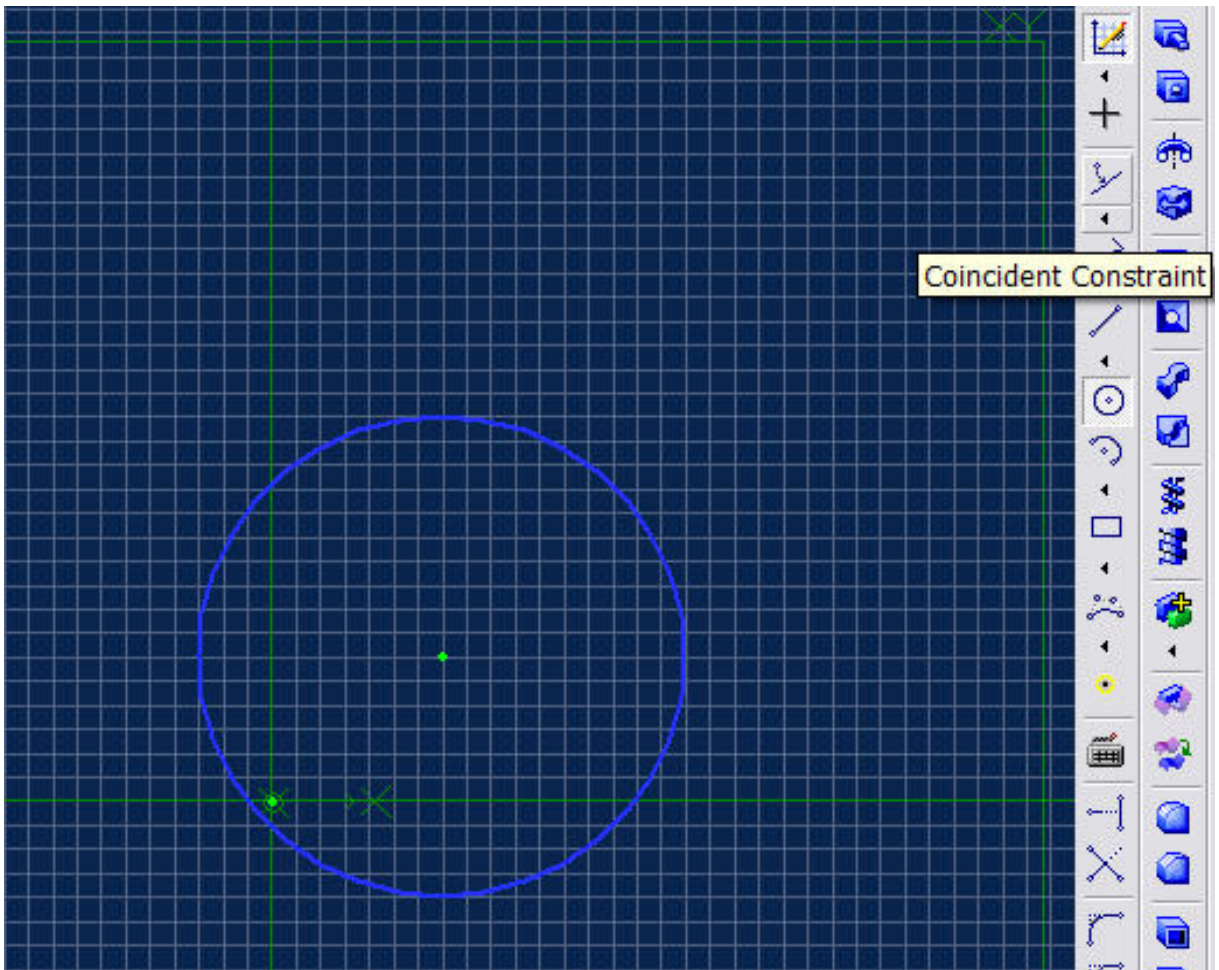


Figure 17

1.11 Modify Instead of Delete

Each sketch element used to create a solid generates a unique identifier associating it to its related solid feature. If this element is deleted, and replaced in the sketch, the solid feature will have a new identifier. The possibility exists that features linked to the old feature identifier will not recognize the new one. See figures 18 – 25)

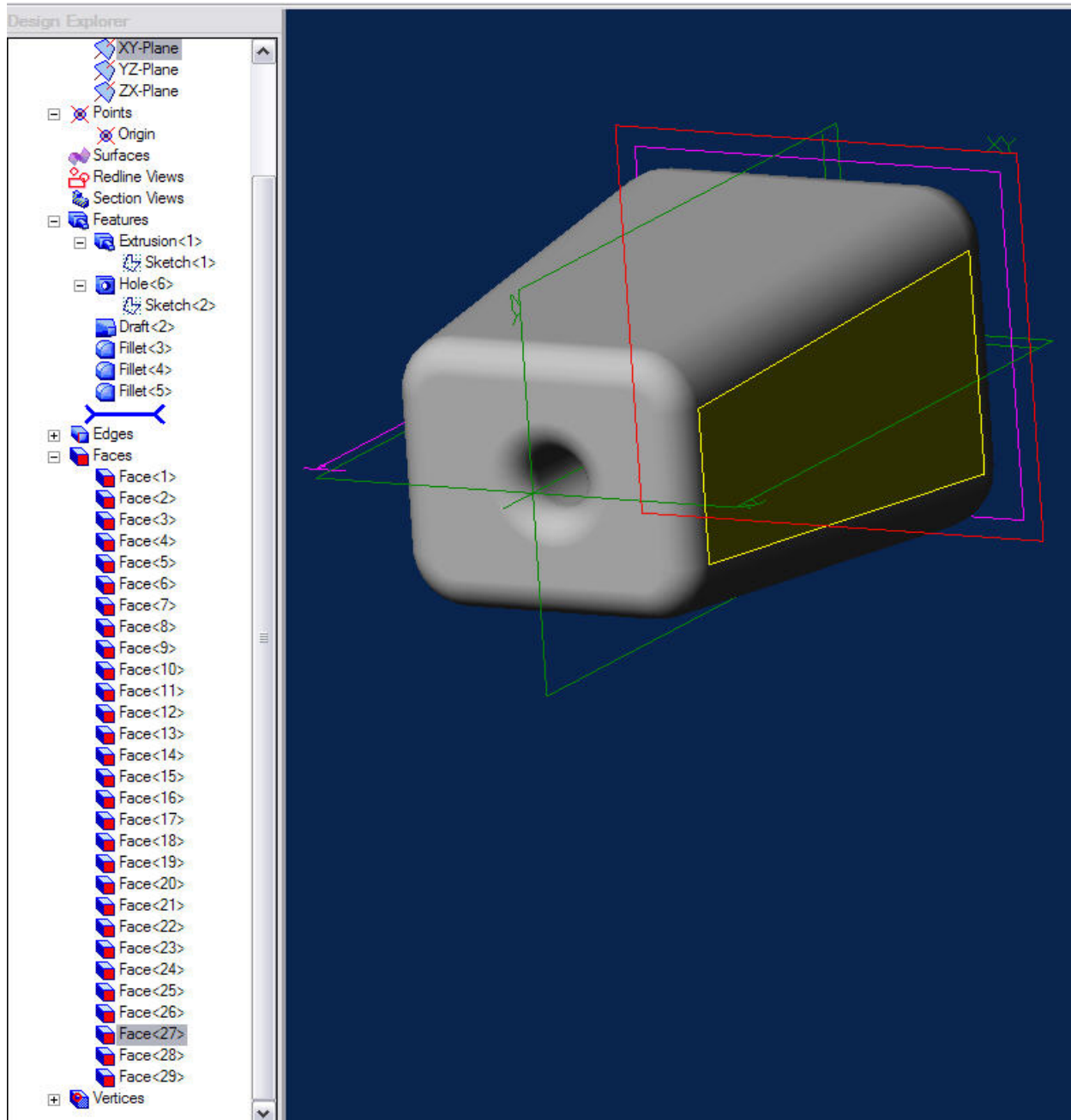


Figure 18

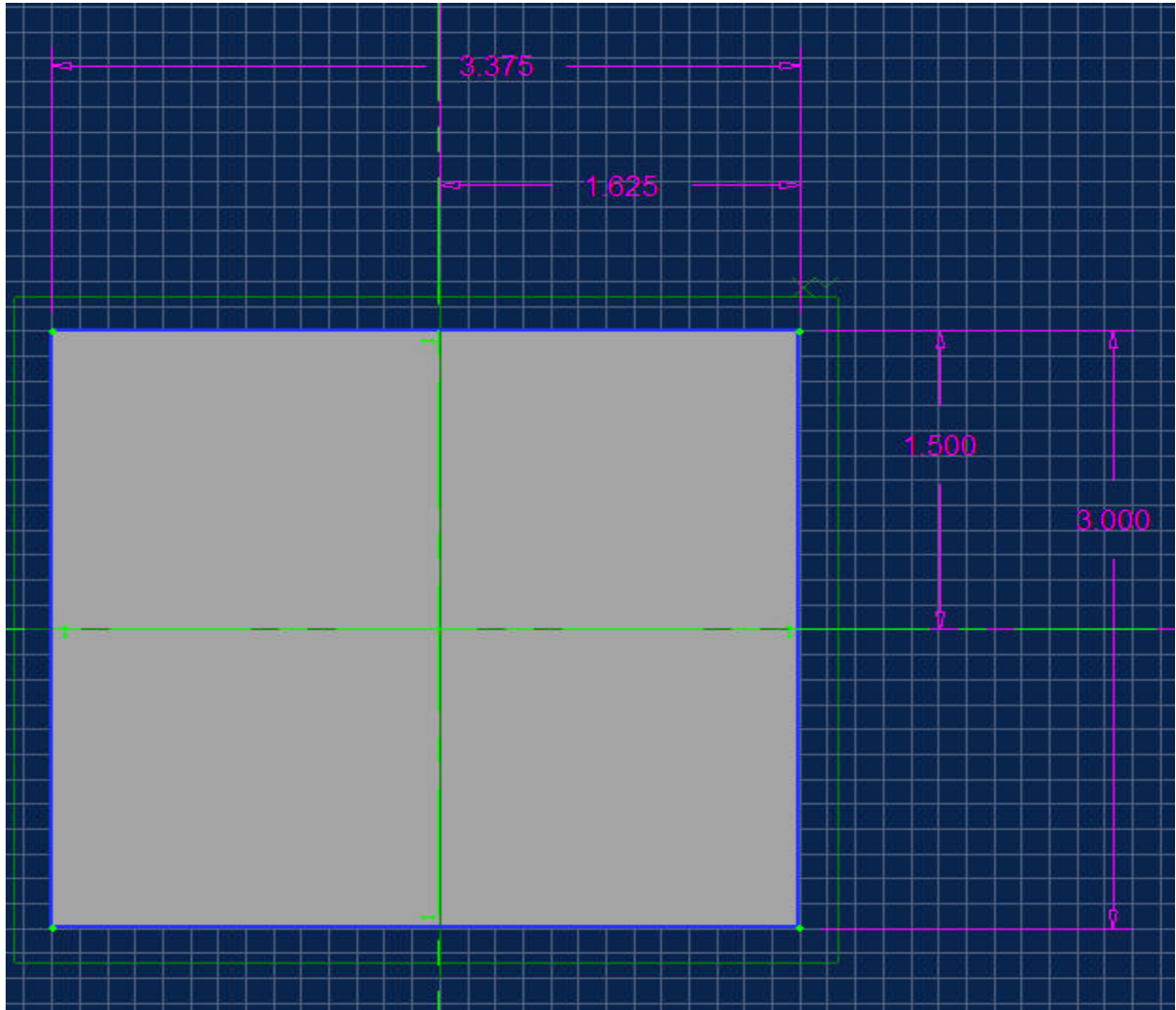


Figure 19

Shown in the picture are the original sketch dimensional constraints. We'll change the 3.000 and 1.500 dimensions to 2.75 and 1.375 respectively.

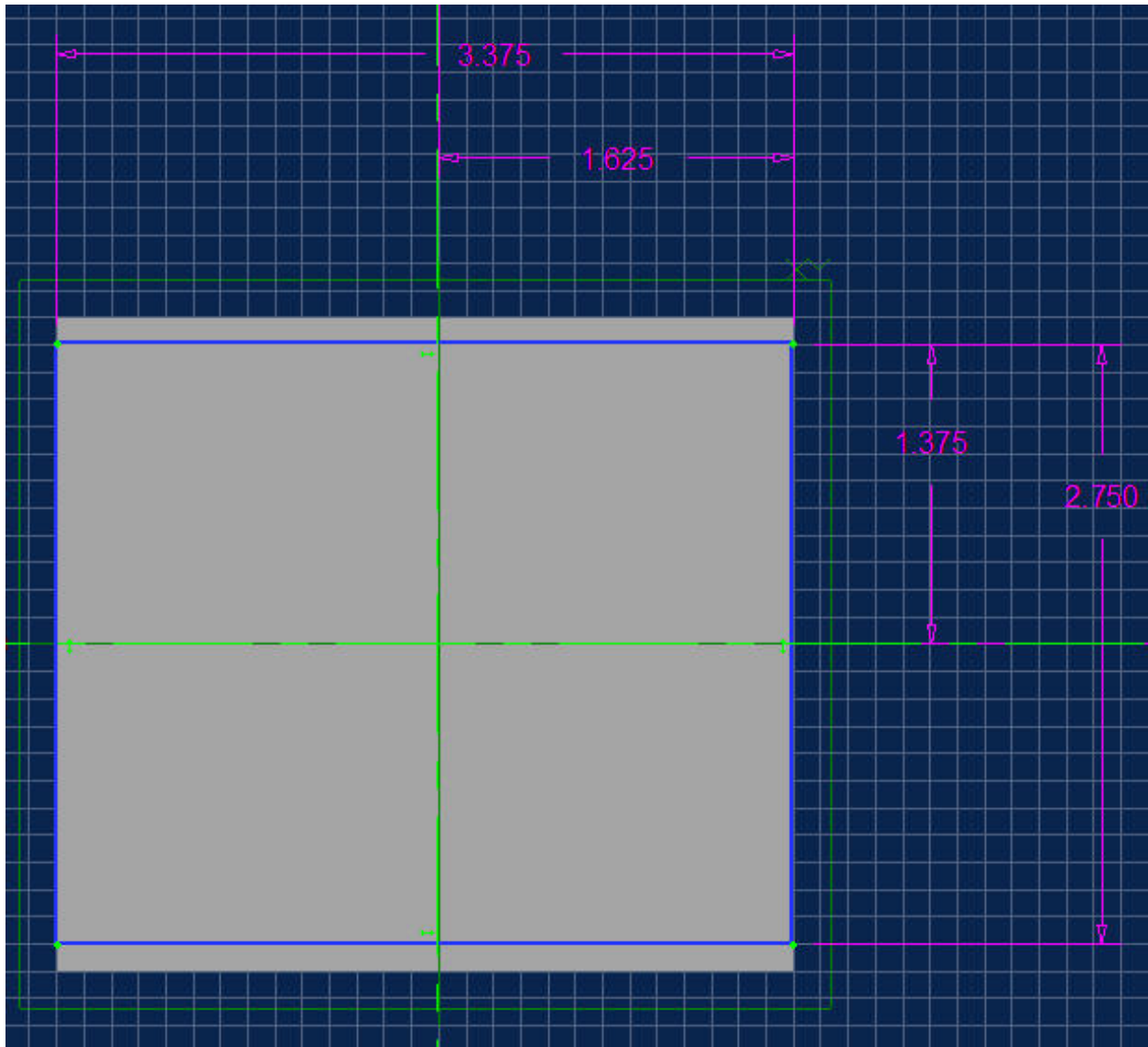


Figure 20

This is the part with the new dimensional constraints in place.

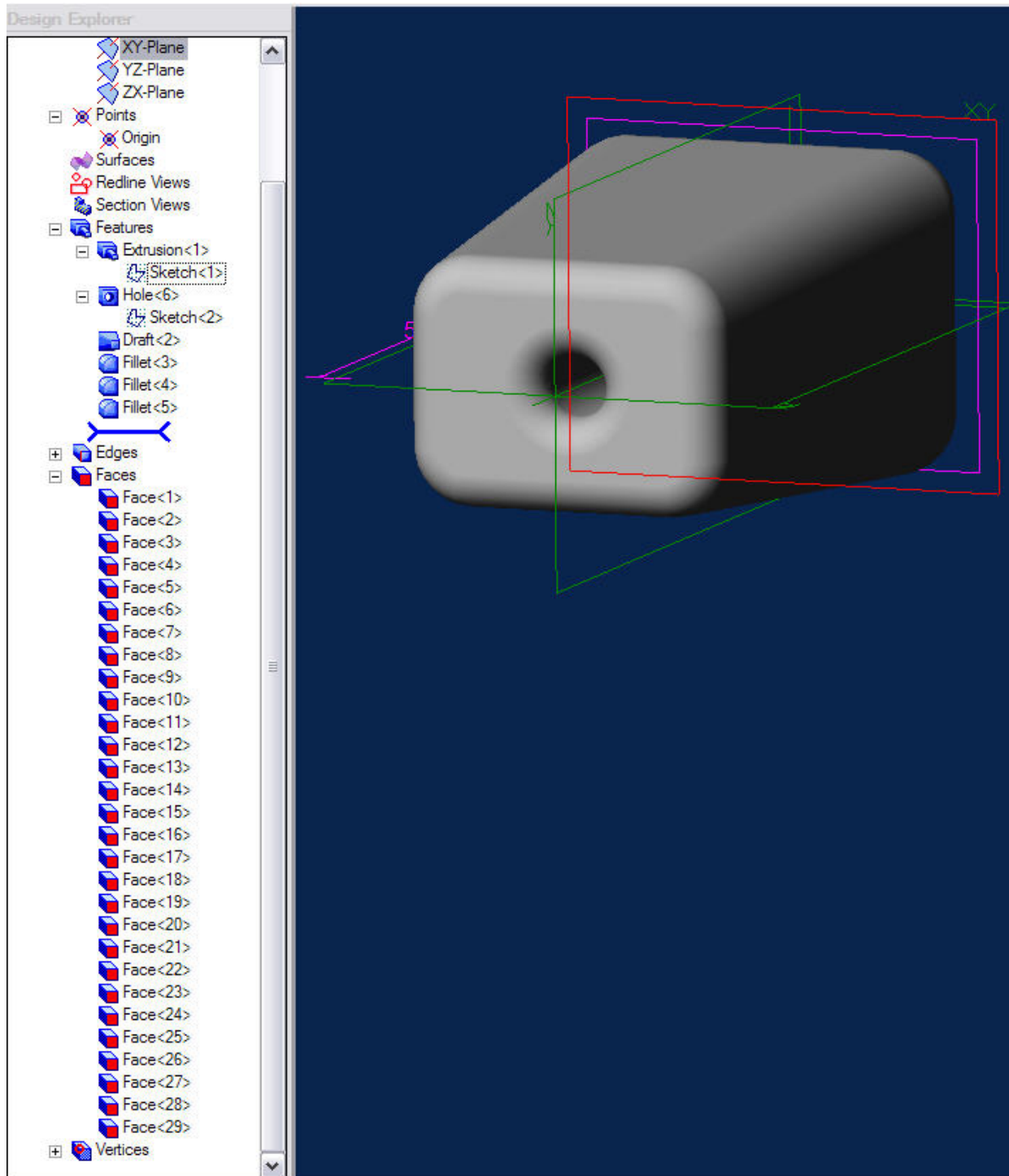


Figure 21

This is the revised part showing the results of the new dimensional constraints. A comparison between the tree structures (figures 18 and 21) will show no change. All identifiers between the sketch elements and resulting solid elements are the same.

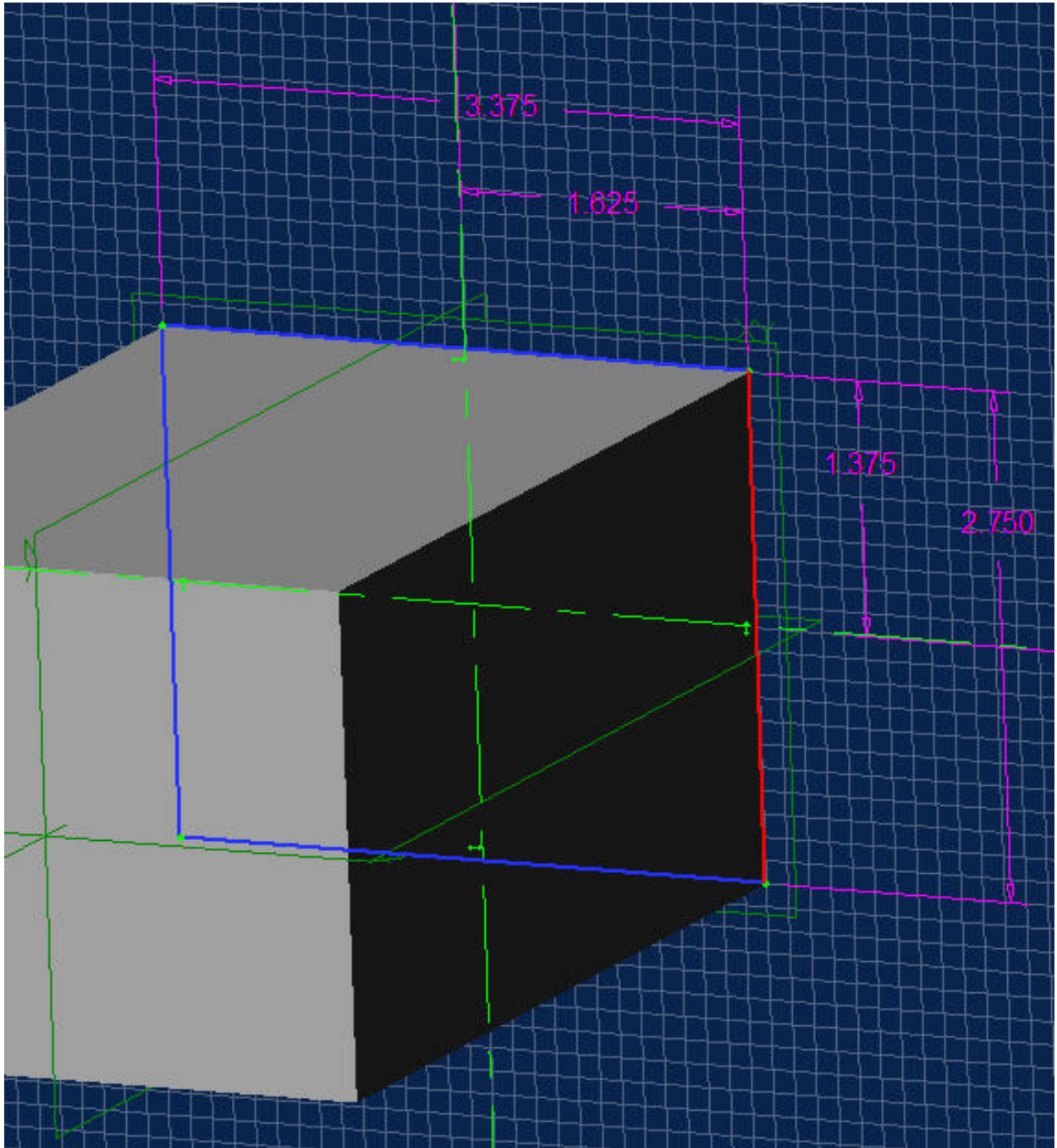


Figure 22

In this view, the vertical member on the right has been marked for replacement (deletion and redrawing).

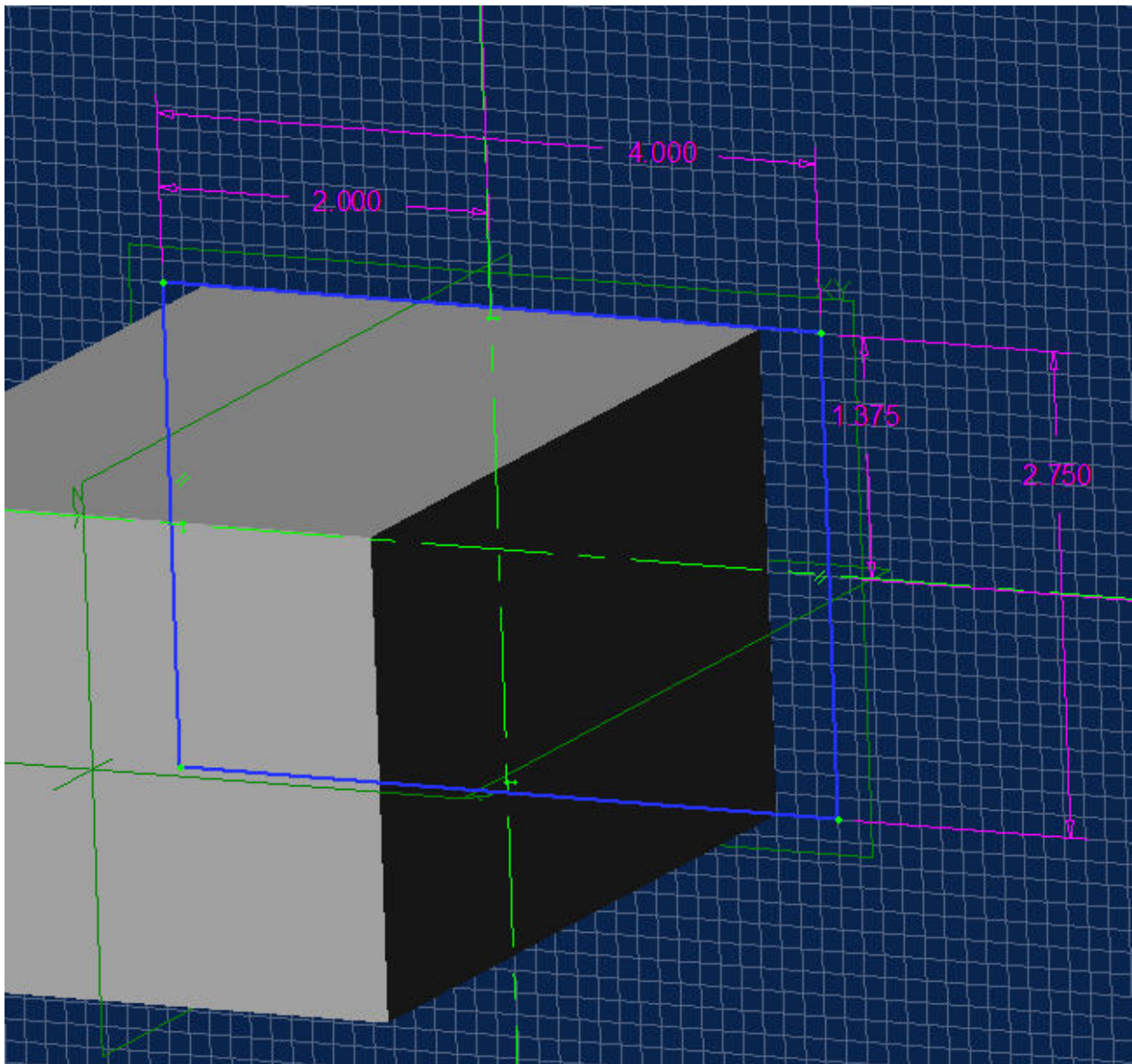


Figure 23

The element has been replaced, and the dimensional constraints associated to it have been replaced (4.000, and 2.000).

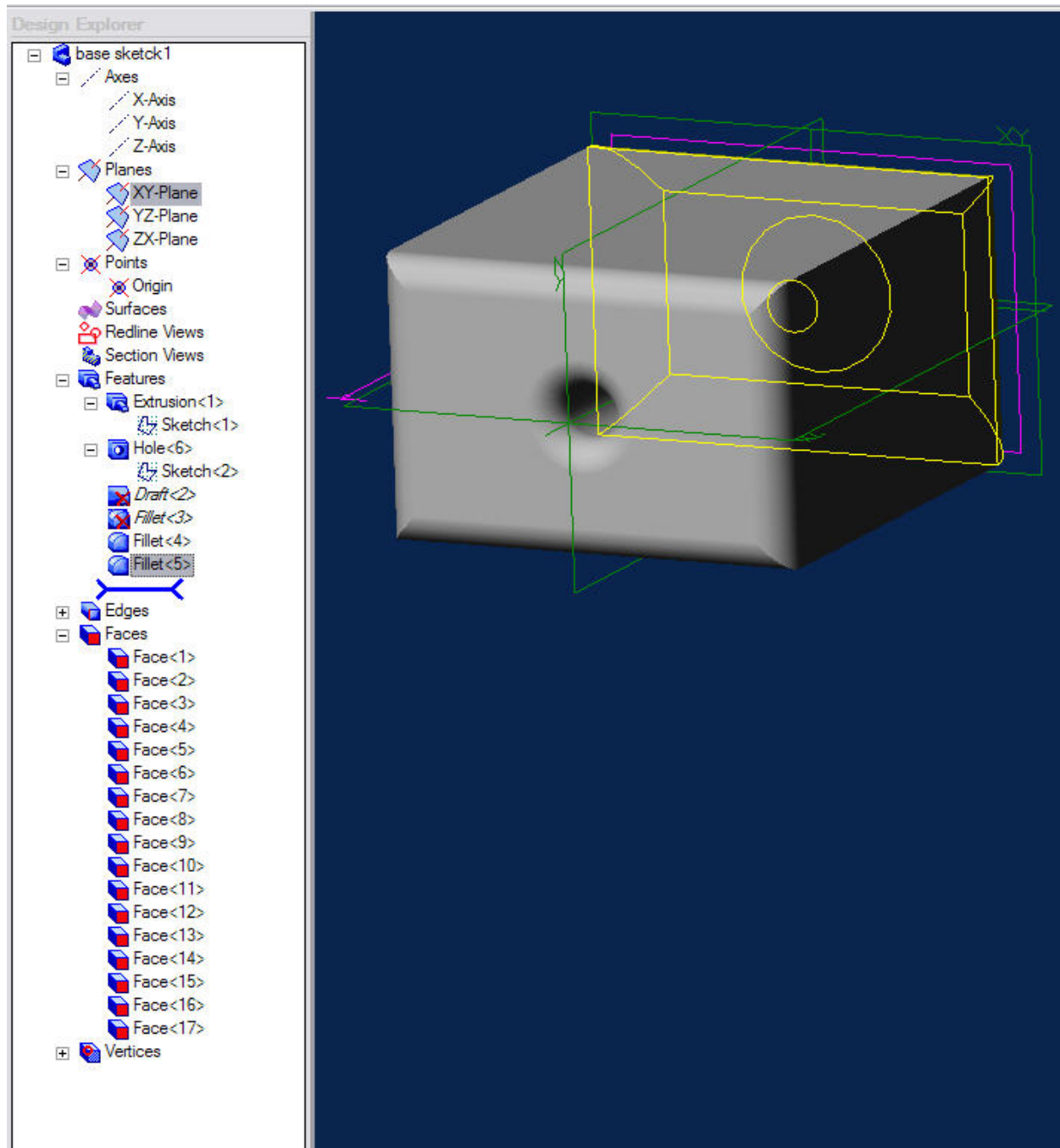


Figure 24

This is the resulting part. Compare the tree structure with that in figures 18 or 21 and note the differences. The drafts and fillets now showing in italics will have to be re-created and there are 11 faces missing associated with those same elements. You can see why the modification of a sketch element is much preferred to replacing it.

1.12 Additional Hints and Tips

Do not use sketched features when standard part design features could be used. For example, use the 'hole' command in the part design toolbar instead of drawing a circle in sketcher and extruding it as a 'cut' feature. This makes editing, re-ordering and deleting in the 3D environment much easier.