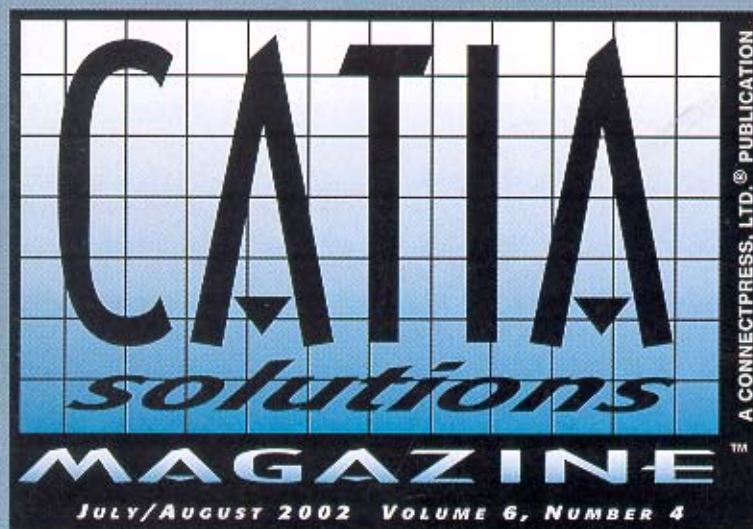


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Basic Part Intelligence

by Richard Cozzens

When I first started using CATIA V5, I was quite impressed with the Sketcher workbench. It seemed to me that it was a vast improvement from my previous Sketcher experience. Creating a profile was simple, and it was just as easy to modify the profile. In fact, it was almost too easy to modify; without constraints the profile could end up relocating randomly. I applied a few constraints, attempting to tie down the profile. It took me some time to be able to control the sketch elements I wanted, without throwing other sketch elements out of whack. At times, it was frustrating—I could see the power in the parametric design, but something was missing. I concluded that what was missing was my full understanding of the process required to create a CATIA V5 part. Instead, I was creating a part that contained information, but not necessarily intelligence. By intelligence, I mean that the entity contains organized information such as its name, its location and its relation to other entities. In V5, the designer is required to assemble and organize the information into something that could be referred to as intelligence. This process is a major shift for those who cut their teeth on CATIA V4.

How well you organize information into intelligence directly affects the time and work required not only to finish the part design but also to update the design. The time spent upfront considering the design intent of the part can save hours of work and frustration down the line. The steps to add intelligence described below are not new, but they are from a different point of view. It is assumed that the reader knows how to create Symmetry Constraints as well as use the Formula tool.

Step One: Stop and take time to analyze the part! This is the easiest step to ignore; it is not required to complete the part yet it is the most critical step in the entire process. You need to be able to answer the following question: How many Sketch features are required to create this part? A manufacturing engineer plans every process required to manufacture a part and a designer should go through the same detail planning when designing a part.

Step Two: Determine the primary or dominate feature. Make this primary feature the Base Feature. The remaining features will be built around the Base Feature.

Step Three: Determine the remaining features requiring a sketch. Remember the fewer number of sketches, the easier the part

is to manage and modify. Use the Dress-Up features when possible (i.e., fillets, chamfers, threads/taps, etc.). Creating these features using the Dress-Up tools will save several steps in the design process and can be quickly created, copied and modified.

Step Four: Answer the following questions: What is the design intent of this part? Is it an individual part? Is the part going into an assembly? Is the part the first of a series of similar parts? Armed with the answers to these questions will help you create and apply the geometrical relationships, driving constraints and functional constraints to control the size and shape of the feature.

Step Five: Constraining Guidelines:

- Use geometric constraints when possible (vertical, horizontal, parallel, etc.).
- Create critical constraints first, then large to small constraint values.
- For remaining constraints, use the Auto Constraint tool. You can go back and quickly delete the unnecessary constraints if needed.

Step Six: Create driven constraints using the Formula or Design Table tools (if applicable).

Step Seven: Name the features in the Specification Tree so that they are easily identifiable. This is listed as Step Seven but actually should be updated as the features are created. (This is a good habit to get into regardless of the simplicity of the part.) By doing this, modifications will be easier for you and others who will be working with the part.

Now let's apply the steps just discussed to a simple part. The part shown in *Figure 1* would not be much of a challenge even for new CATIA V5 users, but adding adequate intelligence might be a bit more challenging. Is there a wrong way to create this part? Probably not, but there is definitely a more efficient method. The following is one of the suggested methods.

Step One: In analyzing this part, I decided I could create it with two sketches—the Base Feature sketch and the Secondary Feature Sketch (labeled in the Specification Tree accordingly). The remaining features will be created using the Dress-up tools.

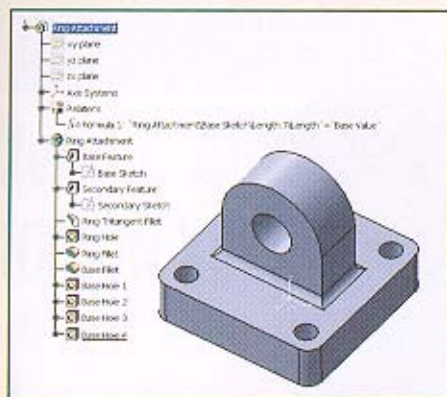


Figure 1.

Step Two: I determined the Base Feature to be the base plate (the flat square). The vertical ring is the Secondary Feature.

Step Three: All five holes require a sketch. The Pattern tool was not used to create the four .375 holes, so the hole-to-edge constraint could be used regardless of the size of the Base Feature. (The fifth hole—the vertical ring—is discussed in Step Two.)

Step Four: Without knowing the true function of this part, it is difficult to determine the design intent. To keep this as simple as possible I have outlined the basic design intent (requirements):

- The Base Feature is to remain square.
- The 0,0,0 point is to remain in the center of the Base Feature, regardless of the size of the base.
- The Secondary Feature (vertical ring) is to remain square (total width equals height) and centered in relationship to the Base Feature, regardless of the size.

Step Five: Figures 2 and 3 show the geometrical constraints used to define the sketch elements. The Symmetry Constraint was created using the Vertical and Horizontal axes. The critical constraints are the width and height values. The width was selected to be the driving constraint and I renamed the parameter Base Value. The height constraint is driven from the width constraint value using the Formula tool; it will always equal the Base Value. In this particular case, there was no need to use the Auto Constraint tool.

Step Six: Both Figures 2 and 3 show driven constraints using the Formula tool. The purpose of these constraints is to keep both sketches perfectly square. The formula makes one side equal to the driven value.

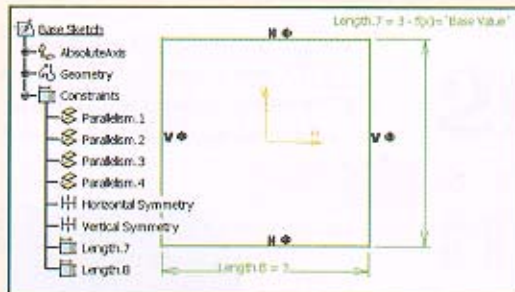


Figure 2.

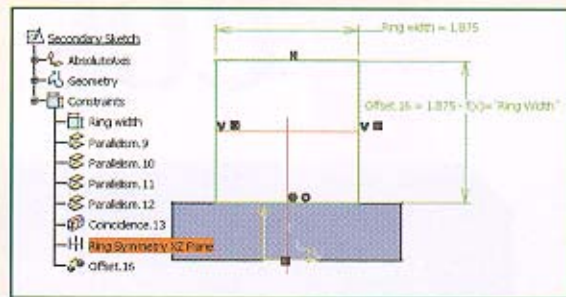


Figure 3.

Step Seven: I updated the Specification Tree as I created the part. I may have gone a little overboard in naming every branch of the tree. My intent in doing so was to help clarify the process.

Does the part contain intelligence or just useless information? Well, let us put it to the test. The intelligence can be tested by doing the following:

1. Go into the Base Sketch and modify the width constraint.
2. Exit the Sketcher Workbench.
3. Update the solid if it is not automatically updated.

If the relationship to all the other features remained the same (still meets the criteria stated in Step Four) then the part has maintained the design intent and the intelligence has been correctly applied. If the design intent was not maintained, you will have to try alternative design and constraining methods until the part maintains the required specifications. Developing this skill takes practice.

Even though this example is a simple part, the process (once learned and understood) can be applied to very complex parts and/or assemblies. I hope I have provided enough details to get started. Additional information and practice parts can be accessed at www.Schroff1.com.

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