Designing with Parametric Sketches

by Cory McConnell

In the world of 3D modeling, one term that comes up frequently is *parametric sketching*. Parametric sketching, the basis for 3D modeling in Autodesk® Inventor™ software, is a type of sketching that uses parameters and equations to drive geometry. Parametric modeling captures the design intent with geometric relationships so if a designer changes one dimension, the entire model updates to accommodate the new parameter. This technology greatly reduces the amount of time designers spend modifying designs and helps get designs to manufacturing faster. This paper explains how parametric sketches are different from 2D drawings and highlights how parametric sketching can improve your design workflow.

What is Parametric Sketching

Depending on a part's form, fit, or function, certain features of the part have quantifiable relationships with other features of the part. For example, holes are a certain distance apart, or an edge is half as long as another edge, and so forth. Parametric sketching provides a method to maintain and take advantage of these relationships in your sketches and in your parts.

Designers use parameters, equations, dimensions, sketch constraints, and construction geometry to define these relationships. For example, placing a dimension between two points creates a relationship between those two points. Because of this relationship, the two points are always that distance apart, even if one of the points is moved. This is a simple example of *design intent*. *(See Figure 1)*

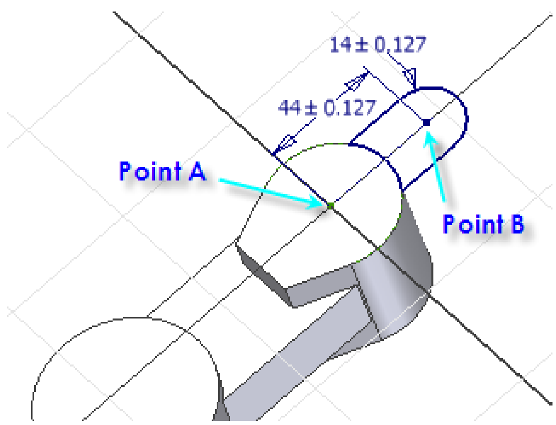


Figure 1
Point B will always be 44 mm from Point A.

The same can be said for sketch constraints. For example, a rectangular feature is centered on a cylindrical feature. Applying a symmetry constraint to the sketch elements ensures that the rectangle is always centered on the cylinder regardless of the size of the rectangle or cylinder. *(See Figure 2)*

Figure 2
Line A is symmetric about the axis of symmetry with Line B.

By projecting geometry from previously created features, you create a relationship between features. For example, by projecting the outer edge of a circular feature and sketching a new circle with a common center point, you relate the new circle to the original circular feature of your part. *(See Figure 3)*

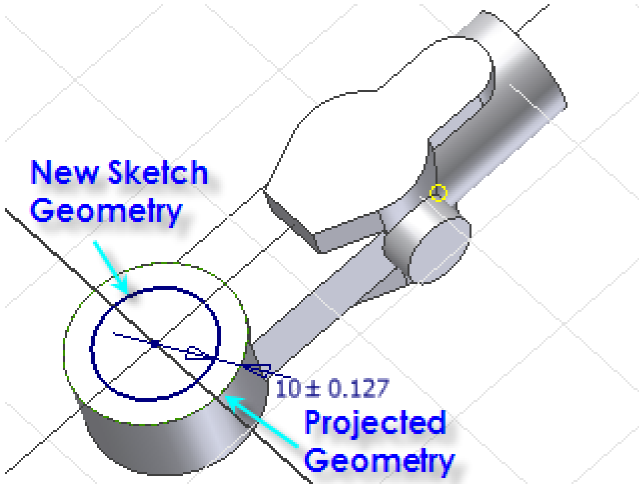


Figure 3
The sketched circle is related to the projected circle.

To use equations to determine relationships, you specify mathematical relationships between dimensions. For example, you specify the length of a rectangle as being twice the width. *(See Figure 4)*

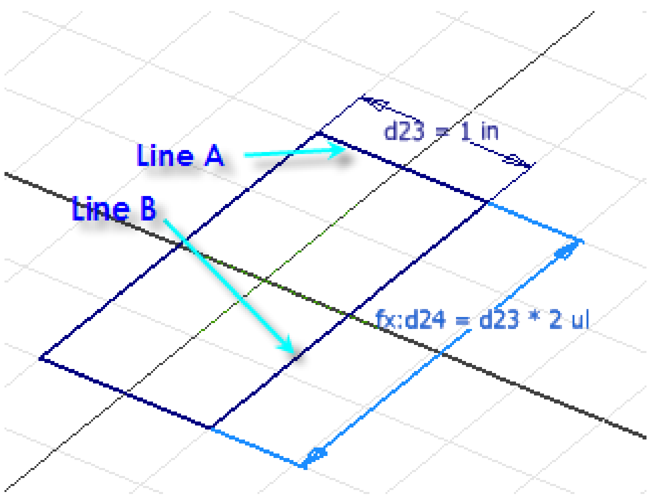


Figure 4
Line B is twice as long as Line A.

In the preceding four examples, the relationships are maintained when the part inevitably changes because of design considerations. Design Intent can be included in your design through the use of these relationships. Parametric modeling gives your parts the ability to maintain your design intent when they are edited by abiding by the relationships that you created in your sketches.

Sketching and Drawing

Parametric sketching and drawing are not the same; each has a different purpose.

A drawing is a collection of geometry (lines, points, and arcs) laid out in a 2D format. These geometric elements, which have no relation to each other, are used to determine the final prints. *(See Figure 5)*

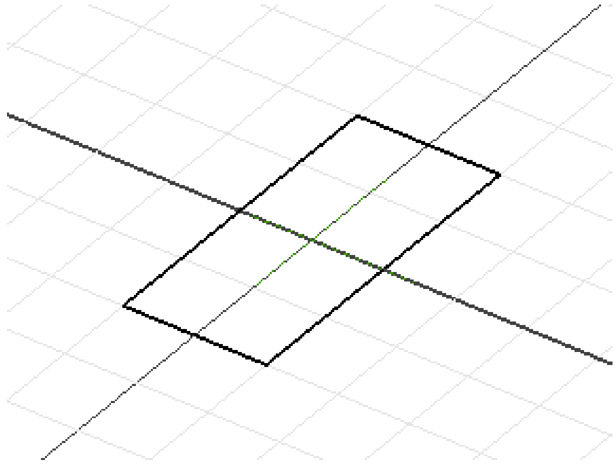


Figure 5
An example of a drawing.

A sketch is a collection of geometry (lines, points, and arcs) coupled with relationships (parameters, equations, dimensions, sketch constraints, and construction geometry) laid out in a 2D format. These geometric elements are related to each other to reflect design intent. These sketches are used to define 3D geometry, which is then projected to 2D for the final prints. *(See Figure 6)*

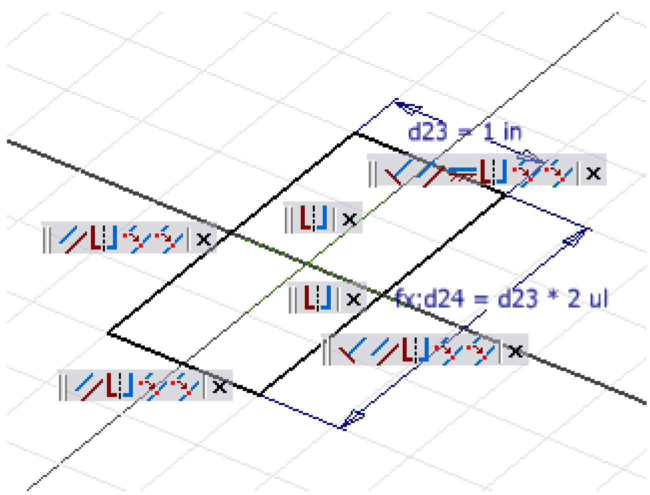


Figure 6
An example of a sketch.

When considering the mechanics of drawing versus sketching, one can conclude that it is probably more efficient to draw a profile than to sketch it. This holds true until it is necessary to make changes to the profile.

For example, to facilitate a design change, it is necessary to change the profile in Figure 5 to be slightly wider. The length must remain twice as long as the width, and the profile must remain centered about the two axes as it is in Figure 5. Although this is possible, it is time-consuming. The same change is required in Figure 6. Because design intent has been captured in the sketch through relationships, changing one dimension will, very quickly, change the profile to satisfy the new width requirement, as well as maintain the original design intent. *(See Figure 7)*

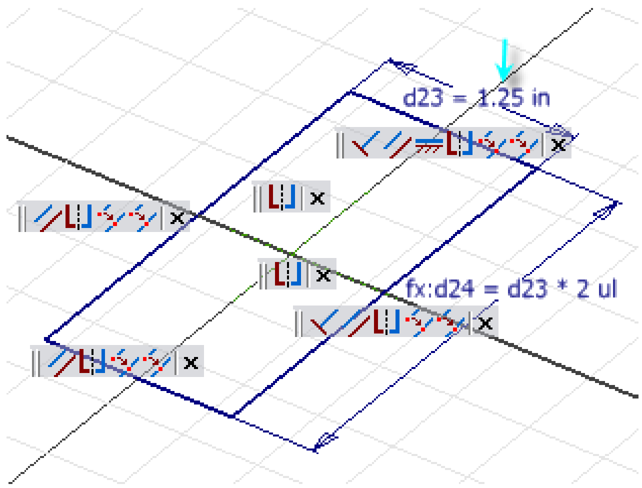


Figure 7
The sketch from Figure 6 with the new width.

Benefits of Parametric Sketches

There are many ways that parametric sketching helps to streamline design changes and iterations, enabling you to realize the downstream benefits.

Parametric sketches reduce the time spent modifying designs. During the design cycle, a part goes through several iterations before a design is finalized. The easier it is to move from one iteration to another, the more design time and overhead are reduced, getting your design to production faster. In the following example, the dimension between the two bores was decreased to accommodate a design change. All the features that are located in relation to the top bore are updated to suit the new location of the bore. *(See Figure 8)*

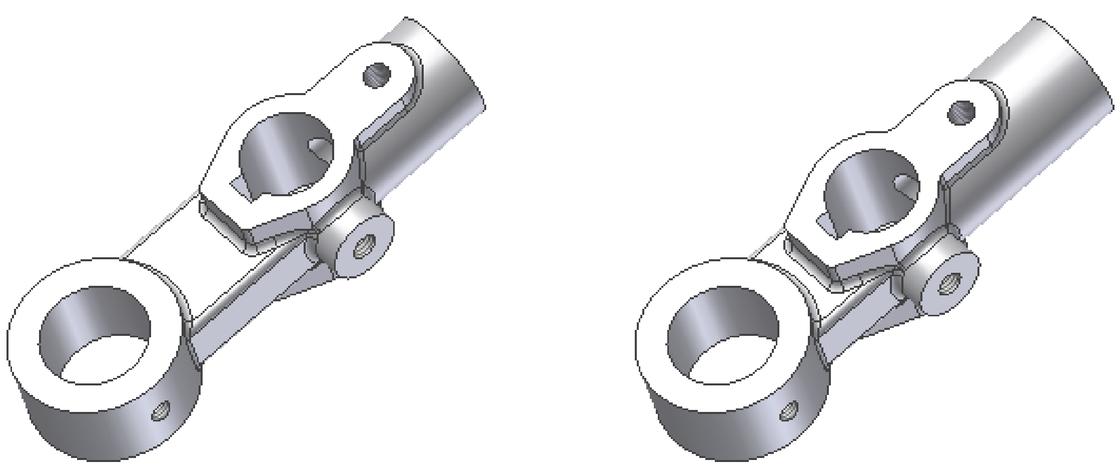


Figure 8

Changing one parametric dimension changes the spacing of the bores, as shown above right.

Sketching parametrically enables you to compare several layouts by applying different dimensions to your design. Generally, there is more than one solution to a design problem. Having the technology to try various layouts for downstream finite element analysis (FEA), motion, physical aesthetics, and so forth, enables you to arrive at functional yet attractive solutions to your design problems much quicker. For example, the following bracket has a cutout in the middle. The size of the cutout is parametrically driven with one dimension. After performing an FEA analysis on the first bracket, it is clear that it is overbuilt. When one parametric dimension in the sketch is changed, the cutout changes to reflect a thinner section. Performing an FEA analysis on the new profile helps determine that it is a more suitable solution. *(See Figure 9)*

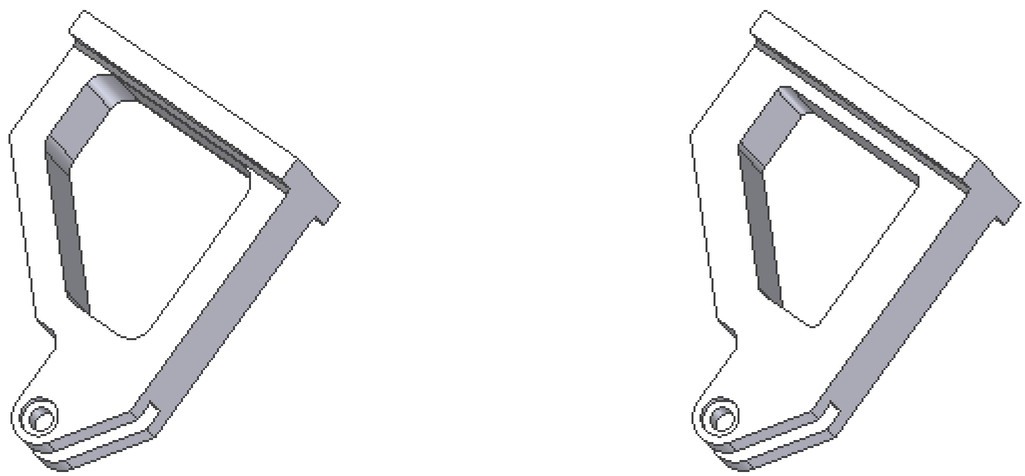


Figure 9

By changing one dimension, the above part can be tested with different profiles, using FEA, to arrive at a light, aesthetically pleasing solution.

Before you spend time on modeling, sketching mechanisms provide testing and confirmation of their intended range of motion. Individual elements of a mechanism are constrained together and dragged, effecting change on adjacent members just as they would in the model or with real parts. In the following figures, a crank-operated accordion system can be analyzed easily by dragging the circle (crank). If it is determined that this mechanism is insufficient to solve certain design criteria, it is simple to modify it or start from scratch with a new concept. *(See Figure 10)*

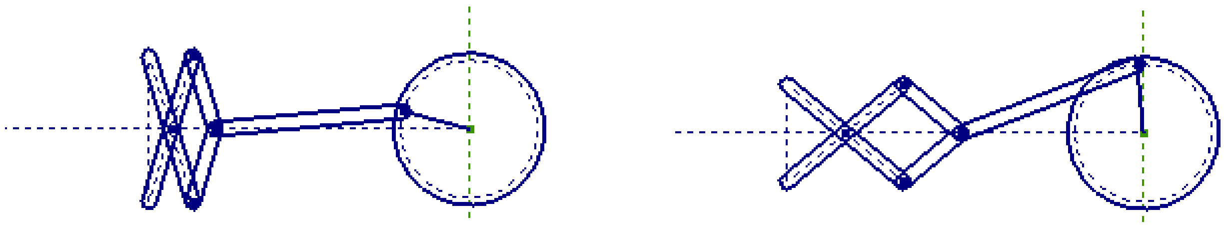
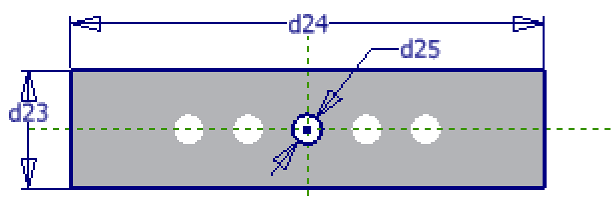


Figure 10
By dragging the point where the long arm attaches to the circle, the mechanism can be tested for displacement.

By adding calculations to your design, you create design rules that help ensure features remain related as the design evolves. In the following example, the hole feature pattern references the overall length of the part in its pattern count and span. The equation specified by “features” determines the number of features, and the equation specified by “span” determines the span of the features. *(See Figure 11)*



Model Parameters		
d23	in	0.5 in
d24	in	2 in
d25	in	0.125 in
d26	in	0.125 in
d27	deg	0 deg
d29	deg	0 deg
features	ul	d24 * 2 ul / 1 in + 1 ul
span	in	d24 - 1 in
User Parameters		

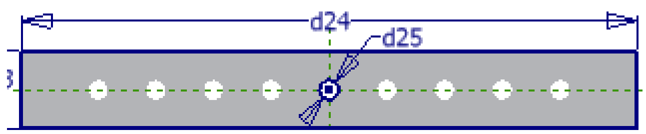


Figure 11

Changing dimension d24 from 2" to 4" effects the above change.

Linking features together parametrically helps to ensure that parts fit together, eliminating the discrepancies that come to light during assembly, when they are most expensive to solve. In the following example, the base part changes size and shape, causing some holes to move. The cover that is fastened with these holes updates, so the cover still fits as it should. *(See Figure 12)*

Original design

Holes are aligned

Original base design

Base design is modified *Cover automatically updates* *Holes remain aligned*

Figure 12

Conclusion

The operating performance, manufacturing cost, and reliability of your design depend on the exact geometric relationships that define the shape and fit of the various parts.

Building this information into your sketches ensures that these important elements of the design are incorporated into the digital model, providing a permanent record of your design intent ensuring that the parts fit together correctly, and giving you more time to work on creating the best design solution.

About the author:
Cory McConnell is a Certified Inventor Expert located in Calgary, Alberta Canada. He owns and operates Mechanix Design Solutions Inc., a mechanical design consulting firm specializing in 3D design and rendering with Autodesk Inventor. www.mechanixdesigns.com