EGR 110
Engineering Graphics
File: N110IL6

Inventor Lecture #6

Reading Assignment:

Read the following in Parametric Modeling with Autodesk Inventor 2008 by Randy Shih:
Chapter 5 – Parametric Constraints Fundamentals

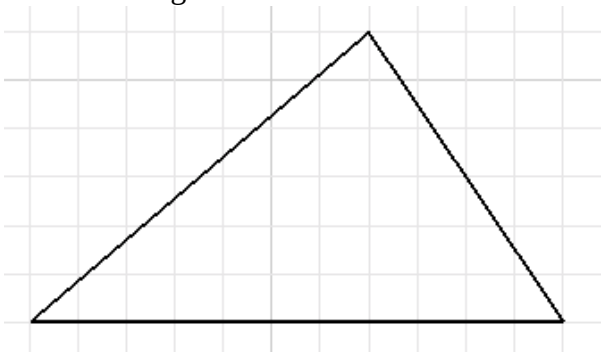
Lecture Outline:

Parametric Modeling (Chapter 5)
Parametric modeling differs from standard CAD software in that it captures the ***design intent*** of the user. It is much more a designer's tool than a draftsman's tool. The series of operations listed in Inventor's browser when a part or assembly is often similar to the series of operations that a machinist might perform in building the part.
Inventor captures the ***design intent*** of the user through:

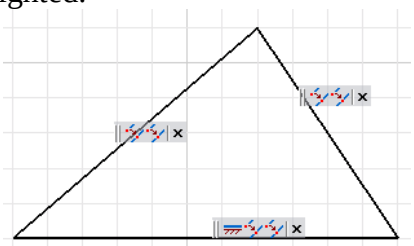
- **Geometric constraints** (such as perpendicular, collinear, tangent, etc.)
- **Dimensional constraints** (controlling the size and location of features)
- **Parametric relations** (user defined mathematical equations composed of dimensional variables and/or design variables) – we will explore these relations today

Geometric and Dimensional Constraints
We will review these constraints plus introduce additional features through an example (similar to the example in Chapter 5 of the text).
Example:

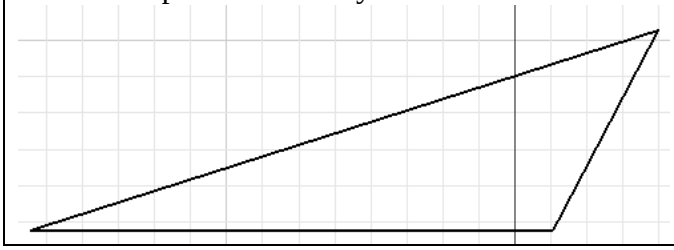
1) Draw a triangle of arbitrary size. The bottom on the triangle should be horizontal.



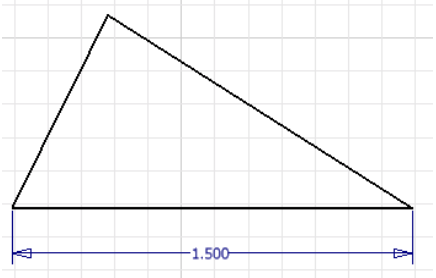
2) Pick Show Constraints from the 2D Sketch Panel and select each line to show the constraints. Move the cursor over each constraint to see which related features are highlighted.



3) Turn off the constraints. Move various sides and points on the triangle. Note that:
   - the bottom line is always horizontal
   - the points are always coincident



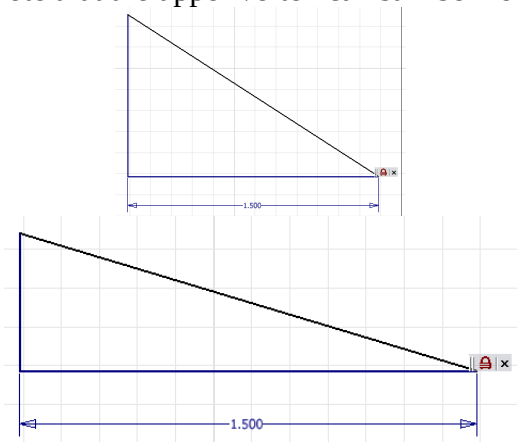
4) Add a dimension to the bottom of the triangle. Note that the bottom length is still fixed and that the entire triangle can still be moved.



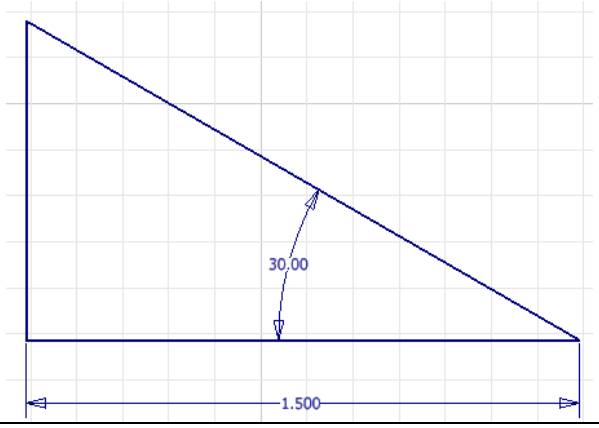
Example (continued)**:**

5) Apply the FIX constraint to the lower right corner of the triangle. Show its constraints. Note that the bottom line changes color as it is fully constrained an can no longer be moved.

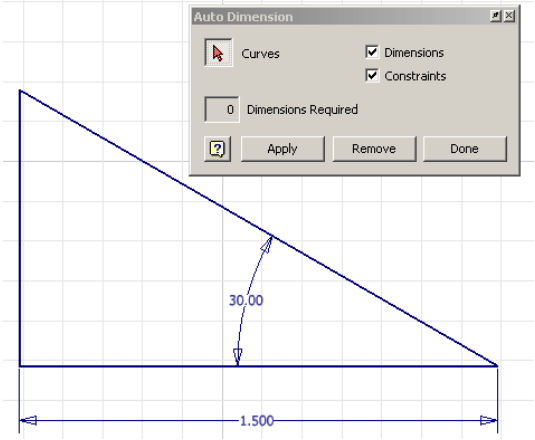
6) Add a VERTICAL constraint to the left line. Note that the upper vertex can still be moved.



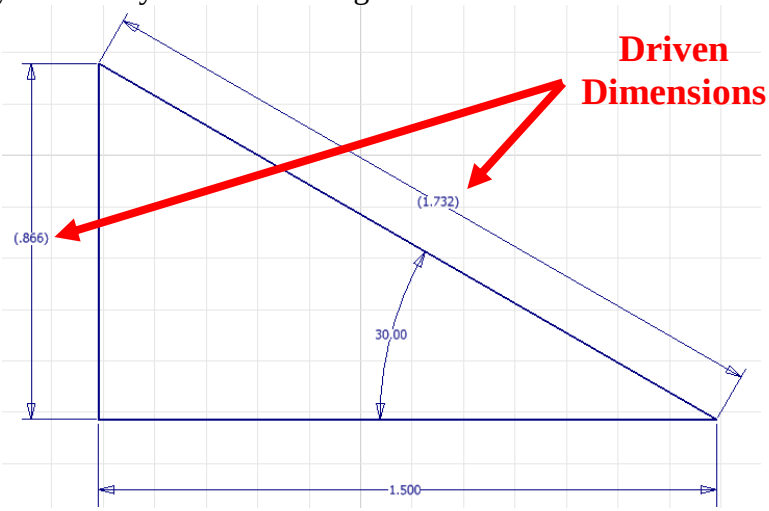
7) Adding an angle dimension will now fully constrain the triangle. (How else could be fully constrain it?) Note that no features on the triangle can be moved now.



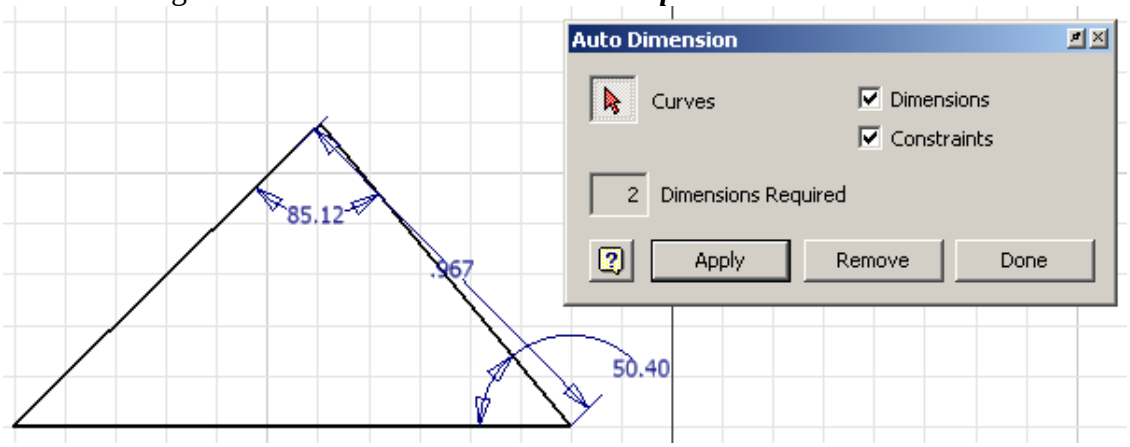
8) Select AUTO DIMENSION from the 2D Sketch Panel. Note that the dialogue box shows that the object is fully constrained.



9) Once an object is fully constrained, additional dimensions are unnecessary. Inventor calls these DRIVEN DIMENSIONS and they will appear in parentheses. You cannot change these dimensions as they are merely calculated using other dimensions in the sketch.

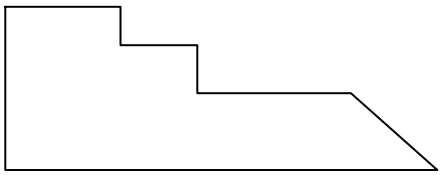


Example: Repeat the last example, but check Auto Dimension after each step. Note how the total number of dimensions needed changes. Try picking Apply in several cases to see which dimensions Inventor will add – it sometimes makes some odd choices! For example, note the results of applying Auto Dimension after Step 1 below. Also note that Inventor will typically still show "***2 Dimensions Required***" after adding dimensions because the object can still be moved. If a point is fixed on the object, then the dialogue box will show "***0 Dimensions Required***."



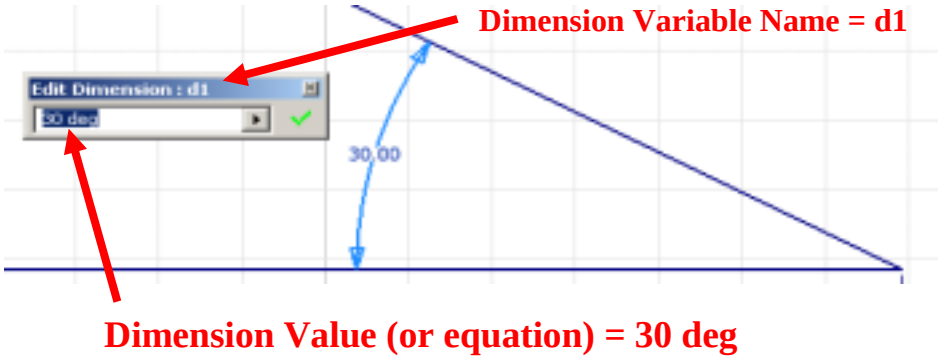
Fully Dimensioning Parts

Parts can be updated more predictably if they are fully dimensioned, although fixing a point isn't crucial. So it is a good practice to check Auto Dimension and see if "***2 Dimensions Required***" is displayed. Try an example:



Parametric Relations

As dimensions are added to a drawing, Inventor assigns a variable name to them in the format "dxx", where xx is a number assigned to each dimension. In the figure below, note that the dimension for the angle has the variable name d1.



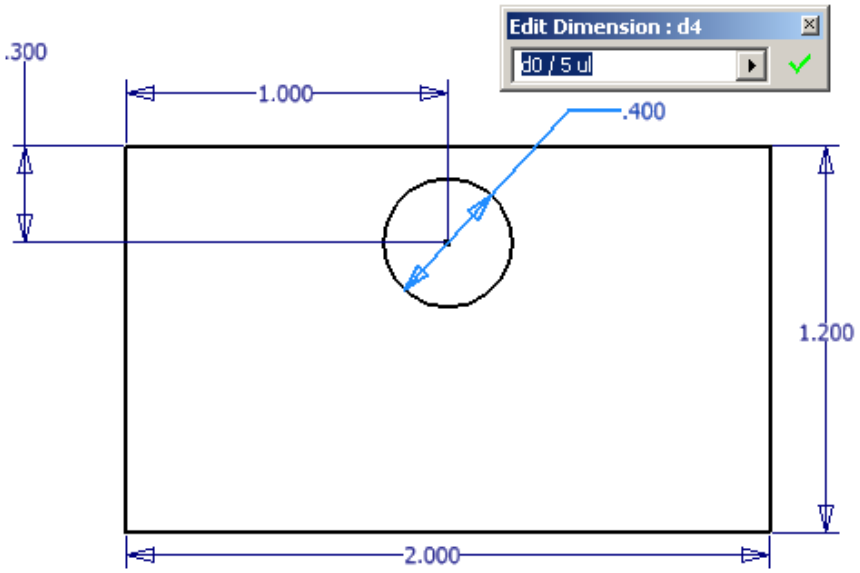
Parametric Equations: A parametric equation is formed when one dimension value (parameter) is expressed in terms of another dimension value (the control parameter). Examples:

d2 = 4*d1
d3 = (d1 + d2)/2
d5 = d4 – 0.125

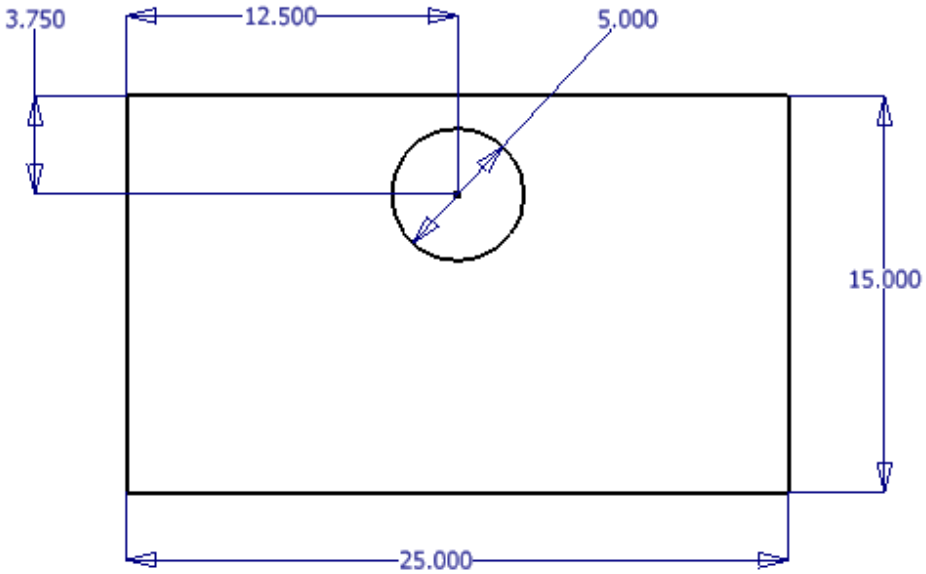
Example:

Create the object shown below subject to the following constraints:

- The base should be 2.000 inches wide.
- The height should be 60% of the base.
- The hole should be centered horizontally.
- The hole should be located a distance from the top equal to 25% of the base.
- The diameter of the hole should be 20% of the base.

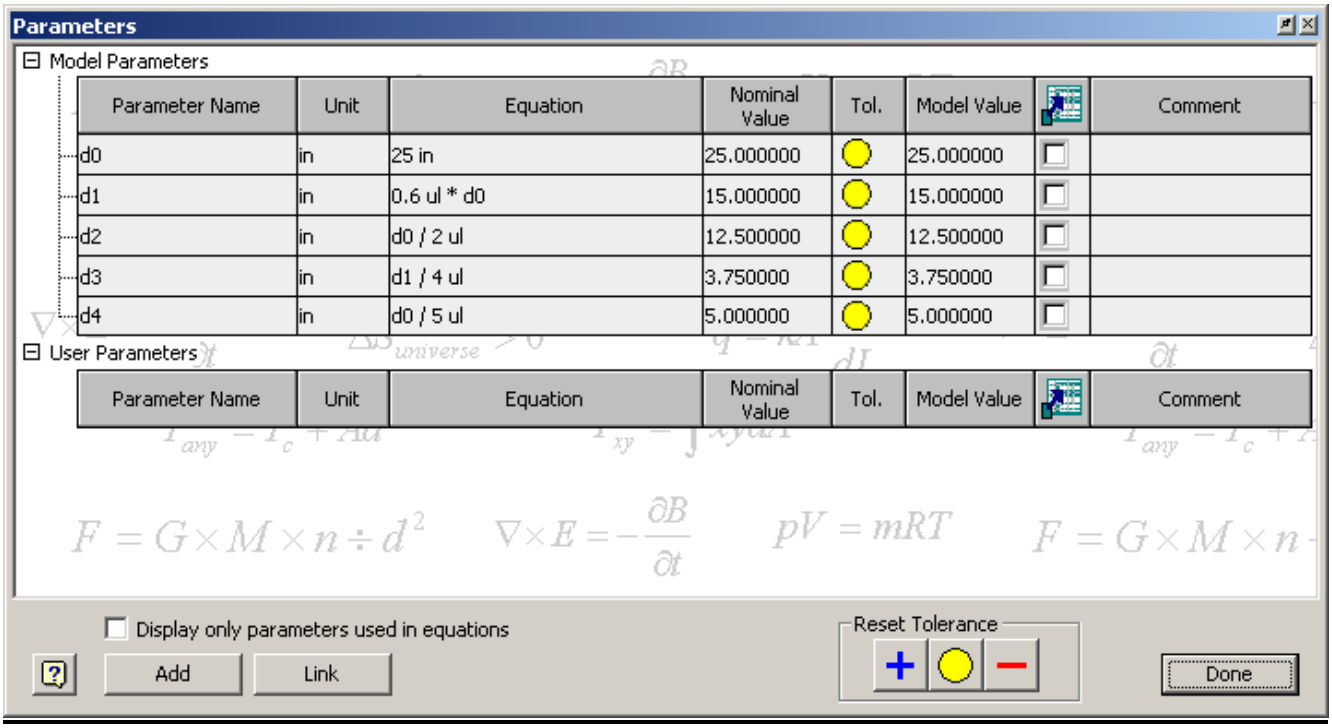


Now change the base dimension to some other values and see that the object changes proportionally.



Viewing Dimension Variables

To view the values (Equations) for each **dimension variable (Parameter Name), select PARAMETERS from the 2D** Sketch Panel.



Notes:

1) Dimensions can also be changed by entering new values or expressions in the **Equation** section above and then selecting **Done**.
2) "ul" means "unitless." Examples:
   - d0/2 ul - divide d0 by a unitless constant 2.
   - d0 + 2.5 mm - add 2.5 mm to d0
   - (100 mm / 6 ul) + 2 in
3) **Model Parameters** (see as d0 – d4 below) are created automatically when you add sketch dimensions, feature parameters (such as extrusion depth), and assembly constraints (such as offsets – covered later). **Reference Parameters** (see d5 below) are automatically created when you create a driven dimension.

10)

12)