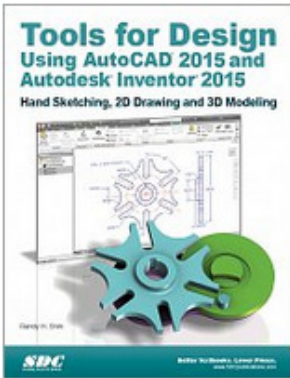


Chapter 10

Parametric Constraints Fundamentals

Learning Objectives:

- ◆ Display, Add, and Delete Geometric Constraints
- ◆ Understand and Apply Different Geometric Constraints
- ◆ Create Fully Constrained Sketches
- ◆ Create Parametric Relations
- ◆ Use Dimensional Variables
- ◆ Display and Modify Parametric Relations

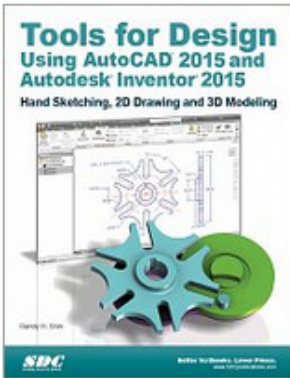


Tools For Design: AutoCAD® & Autodesk Inventor

Introduction

A primary and essential difference between parametric modeling and previous generation computer modeling is that parametric modeling captures the *design intent*.

In the previous lessons, we have seen that the design philosophy of “*shape before size*” is implemented through the use of *Autodesk Inventor's* Profile and Dimension commands



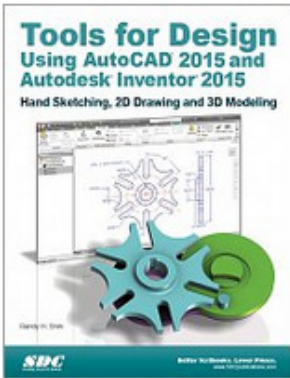
Tools For Design: AutoCAD® & Autodesk Inventor

Introduction

In performing geometric constructions, dimensional values are necessary to describe the **SIZE** and **LOCATION** of constructed geometric entities.

Besides using dimensions to define the geometry, we can also apply geometric rules to control geometric entities.

More importantly, *Autodesk Inventor* can capture design intent through the use of **geometric constraints**, **dimensional constraints**, and **parametric relations**.



Tools For Design: AutoCAD® & Autodesk Inventor

Fully Constrained Geometry

In *Autodesk Inventor*, as we create 2D sketches, geometric constraints such as *horizontal* and *parallel* are automatically added to the sketched geometry.

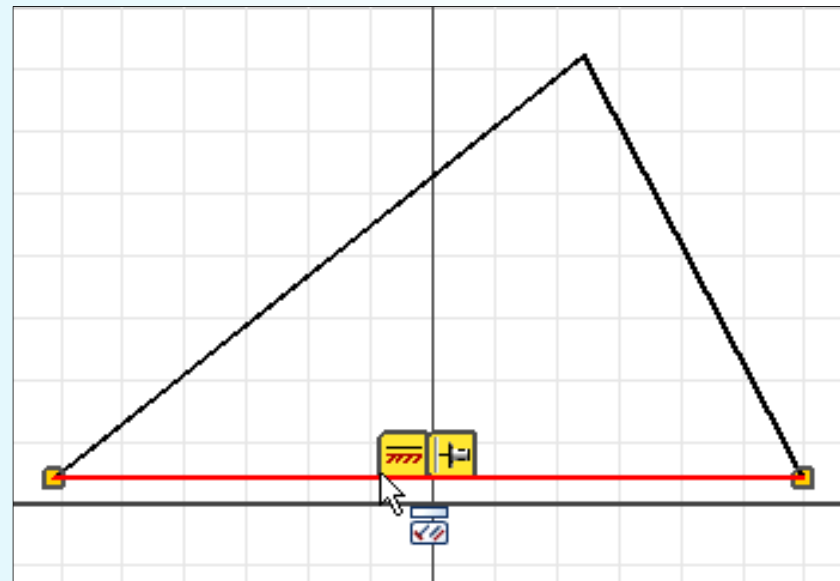
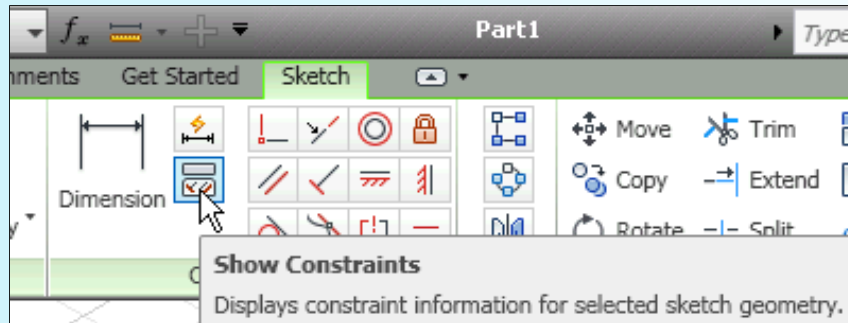
In most cases, additional constraints and dimensions are needed to fully describe the sketched geometry beyond the geometric constraints added by the system.

Although we can use *Autodesk Inventor* to build partially constrained or totally unconstrained solid models, the models may behave unpredictably as changes are made.

In most cases, it is important to consider the design intent and to add proper constraints to geometric entities.



Show Constraints

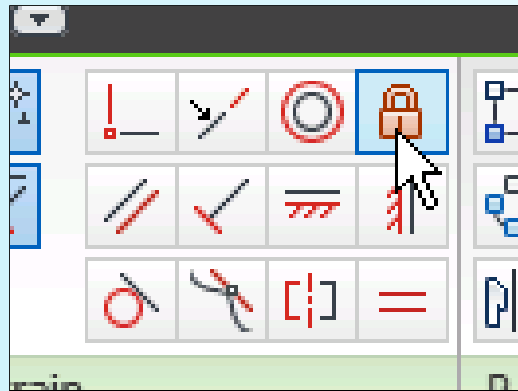


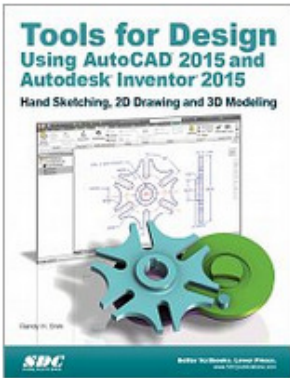
Apply Geometric Constraints

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Hand Sketching, 2D Drawing and 3D Modeling



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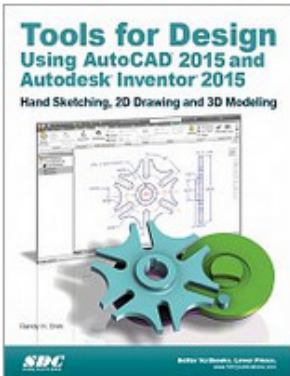




Tools For Design: AutoCAD® & Autodesk Inventor

Over-constraining and Driven Dimensions

We can use *Autodesk Inventor* to build partially constrained or totally unconstrained solid models. In most cases, these types of models may behave unpredictably as changes are made. However, *Autodesk Inventor* will not let us over-constrain a sketch; additional dimensions can still be added to the sketch, but they are used as references only. These additional dimensions are called ***driven dimensions***. *Driven dimensions* do not constrain the sketch; they only reflect the values of the dimensioned geometry. They are enclosed in parentheses to distinguish them from normal (parametric) dimensions. A *driven dimension* can be converted to a normal dimension only if another dimension or geometric constraint is removed.



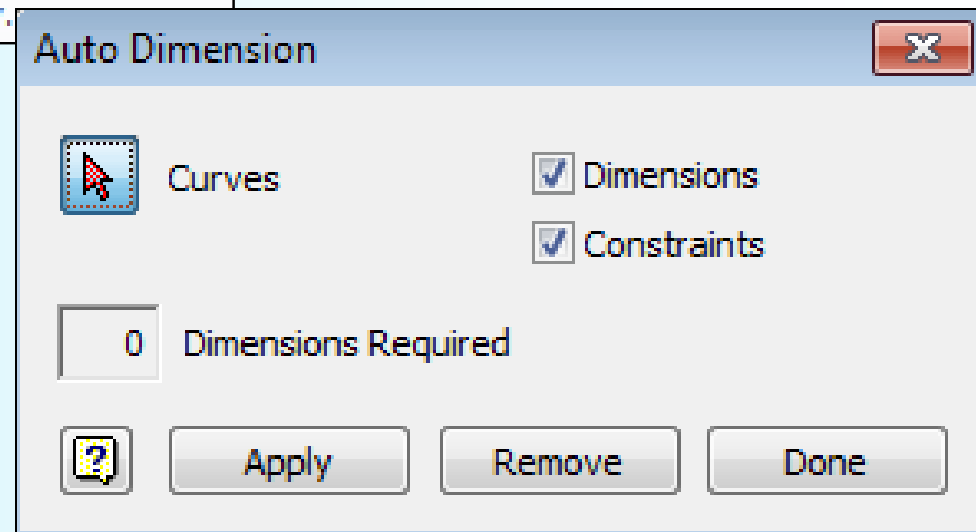
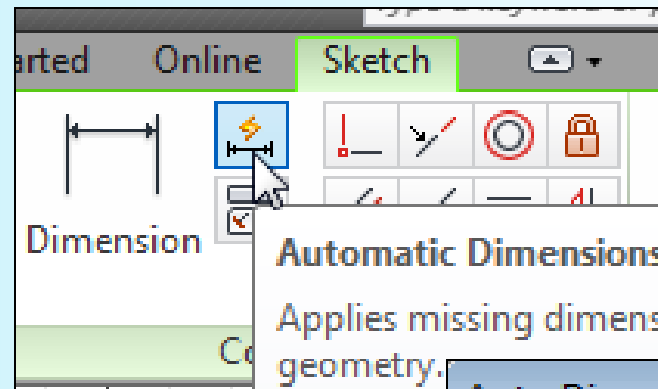
Tools For Design: AutoCAD® & Autodesk Inventor

Auto Dimension

In *Autodesk Inventor*, the **Auto Dimension** command can be used to assist in creating a fully constrained sketch. Fully constrained sketches can be updated more predictably as design changes are implemented. The general procedure for applying dimensions to sketches is to use the **General Dimension** command to add the desired dimensions, and then use the **Auto Dimension** command as a way to check if additional dimensions are needed to fully constrain the sketch. The Auto Dimension command can also be used to apply some additional dimensions that are needed. It is also important to realize that different dimensions and geometric constraints can be applied to the same sketch to accomplish a fully constrained geometry.



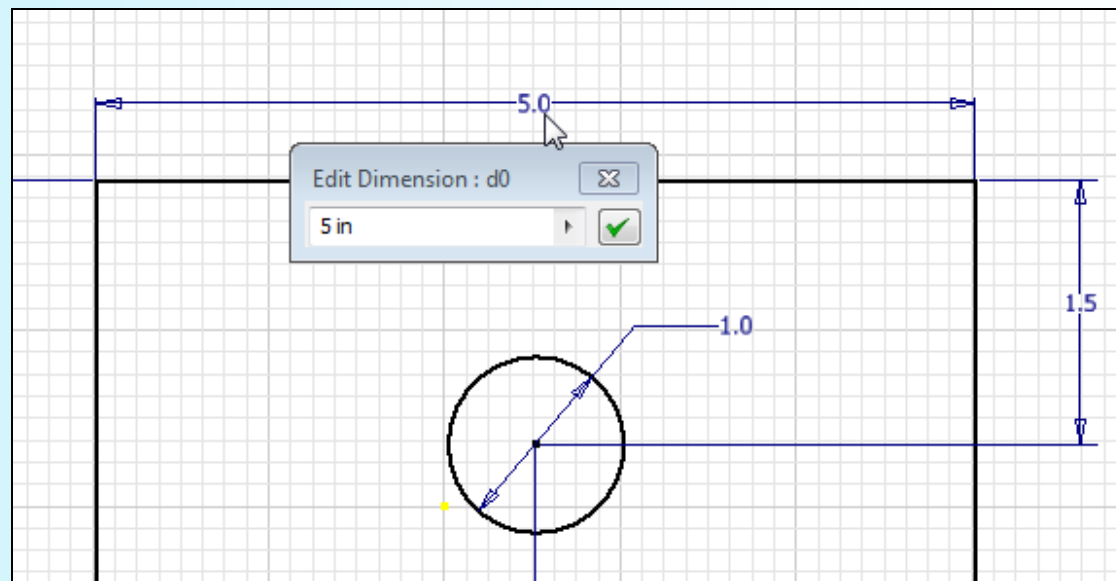
Auto Dimension





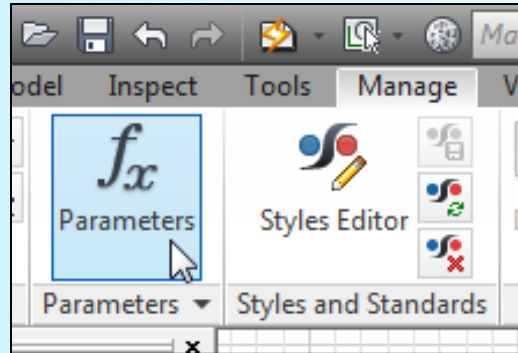
Dimensional Values and Dimensional Variables

Initially in *Autodesk Inventor*, values are used to create different geometric entities. The text created by the Dimension command also reflects the actual location or size of the entity. Each dimension is also assigned a name that allows the dimension to be used as a control variable. The default format is “dxx,” where the “xx” is a number that *Autodesk Inventor* increments automatically each time a new dimension is added.



Parameters and Relations

Tools for Design
Using AutoCAD 2015 and
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Hand Sketching, 2D Drawing and 3D Modeling



Parameters

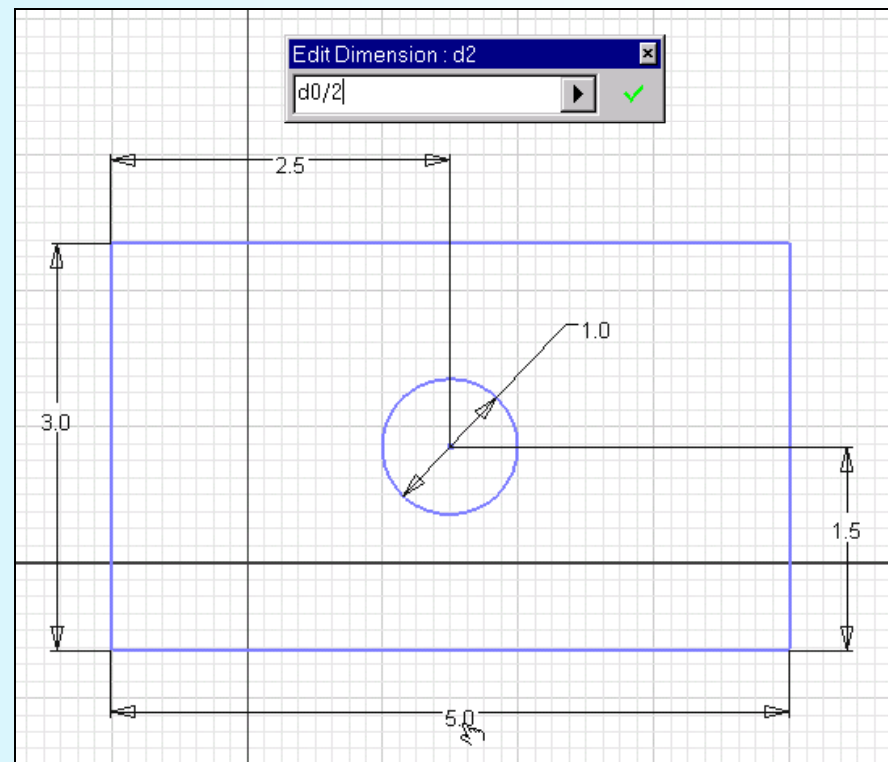
Parameter Name	Unit/Typ	Equation	Nominal Value	Tol.	Model Value	Key		Comment
Model Parameters								
d0	in	5 in	5.000000	●	5.000000	<input type="checkbox"/>	<input type="checkbox"/>	
d2	in	d0 / 2 ul	2.500000	●	2.500000	<input type="checkbox"/>	<input type="checkbox"/>	
d3	in	3 in	3.000000	●	3.000000	<input type="checkbox"/>	<input type="checkbox"/>	
d4	in	d3 / 2 ul	1.500000	●	1.500000	<input type="checkbox"/>	<input type="checkbox"/>	
d5	in	1 in	1.000000	●	1.000000	<input type="checkbox"/>	<input type="checkbox"/>	
User Parameters								

Update
 Immediate Update
 Reset Tolerance:



Parametric Relations

In parametric modeling, dimensions are design parameters that are used to control the sizes and locations of geometric features. Dimensions are more than just values; they can also be used as feature control variables.



Using the Measure Tools

Besides using the Measure tools to get geometric information at the 2D level, the measure tools can also be used on 3D models.

