

IM462304

# Inventor Sketching: Beyond the Basics

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## Learning Objectives

- Discover sketching tips and tricks
- Learn how to effectively use constraints
- Learn about Inventor Smart Dimensioning
- Learn how to manipulate geometry while sketching.

## Description

Sketching is the basis of any model. In this class, we'll explore sketching within Inventor software, give you the time-saving tips and tricks to make you more productive, and show you how to build a rock-solid foundation for your models.

## Speaker(s)

I am the Technical Services Manager at Prairie Machine, a mining equipment manufacturer. Reporting to the general manager, I am responsible for overseeing the company's technical operations and the strategic technical growth.

I have been using AutoCAD since r13, I cut my solid modeling teeth on Mechanical Desktop, and have been using Inventor since inception.

## Sketching

Most 3D models start from a sketch, then further defined through sketched and placed features. Sketches contain 2D or 3D objects. Use the Inventor Sketch environment to sketch, constrain, and dimension 2D geometry.

**Sketching is the foundation** for most of the components modeled. So it is important to have strong sketching skills and to build proper sketches

Where did I get these tips, tricks, and suggestions?

Firstly from my many years of using Inventor, both from using the software and from watching/learning from other users. The old saying is “*it’s not what you know, it’s who you know*”, but it’s really both. I am always on the lookout for any time saving steps that reduce the time to get things done or will build stronger, better models. So, here are some of my favorites.

From Inventor’s help...

*Part models created in Autodesk Inventor start with sketches, which you create by drawing geometric elements such as points, lines, shapes, and arcs. The sketch becomes the basis for sketched features, such as extrusions, revolutions, lofts, coils, or sweeps, which add volume to the sketched part.*

*When starting a sketch, you can specify a planar face, work plane, or sketch curve.*

*A sketch profile is a closed loop defined by sketched or reference geometry that represents a cross-section of a feature. An open profile defined by sketched segments, arcs, or splines can define a surface shape or extend to boundaries to close a region.*

## Getting Your Sketch Started

### Using Sketches to define Work Features

**Offset Workplanes while Sketching** - generate an offset workplane, with a sketch, in one-step, by dragging off the desired face with the create sketch command active.

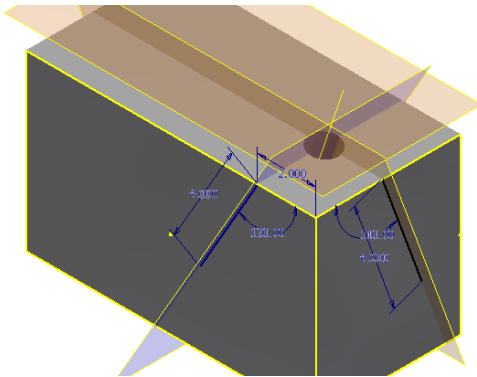


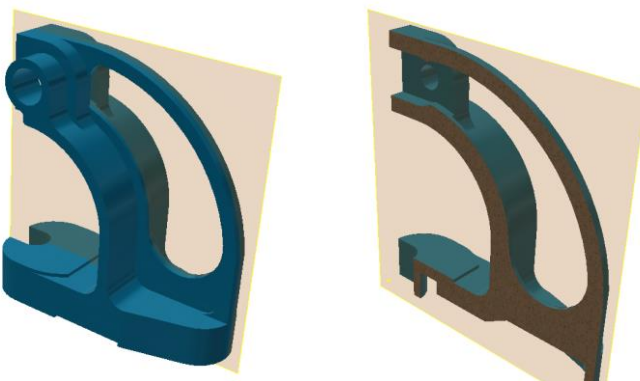
Figure 1: Work Features from Sketch Geometry

### Use Sketches to Define Work Features

Sketch geometry, especially lines can provide an easier method for creating difficult to locate work features.

### Slice Graphics

When the model is in the way, use **Slice Graphics** to remove everything between you and the sketch plane.



Initiate **Slice Graphics** via:

- the right-click menu
- by pressing F7
- the option in the Status bar



If sliced in the wrong direction, turn off the slice graphics, rotate the model, and try slice graphics again.

### Copying & Pasting

Using the **Windows Clipboard**, create copies of sketch geometry pasted into the same sketch, into the same model, or even between parts. Use the standard Microsoft Windows functions right-click Copy or <Ctrl> + C and right-click Paste or <Ctrl> + V

## Project Geometry

Use **Project Cut Edges** to associatively project edges of the model that intersect with the sketching plane. Think of it as the edges of a section view and your sketch is the section line

**Project Flat Pattern** unfolds a disjointed face or faces into the sketch plane.

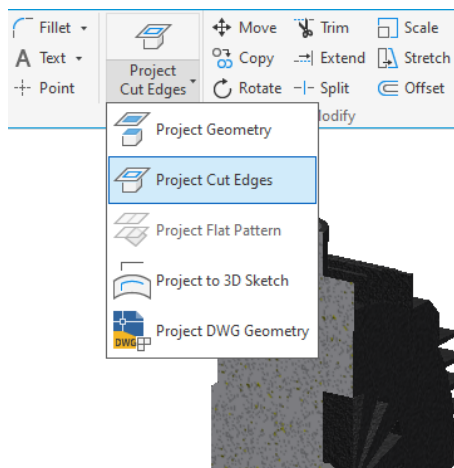


Figure 2: Project Cut Edges

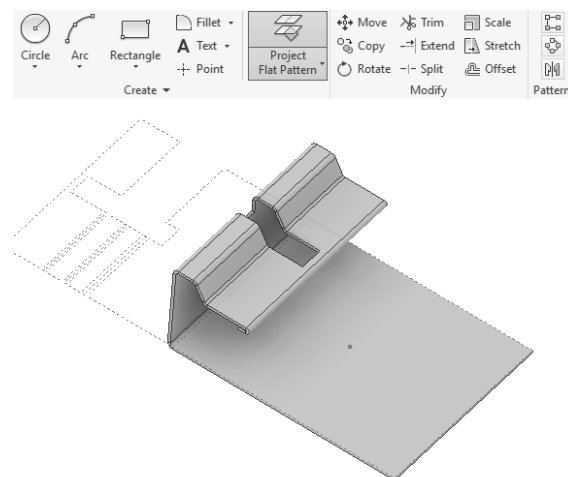


Figure 3: Project Flat Pattern



**Project Flat Pattern** is great for the situations where you do not require the overhead and advanced options of the Sheet Metal Unfold / Refold features. It is for when you just want to reference existing geometry edges

## AutoCAD Geometry

**Importing AutoCAD Geometry** - why redo when you can reuse? AutoCAD geometry can be imported into your sketches. Once it is imported, add dimensions and constraints as if the geometry originated in Inventor.



You can copy & paste directly from AutoCAD into Inventor to bypass the wizard.

With the geometry imported, use **Auto Dimensioning** to apply the “easy” constraints like horizontal, vertical, perpendicular, and parallel.

Utilize **AnyCAD** to associatively import AutoCAD drawings into your Inventor model. Then, from within a sketch, use Project DWG Geometry to project in reference objects from the AutoCAD drawing. If the AutoCAD drawing changes, this projected geometry automatically updates.

## Make Project Geometry Work for You

By default, projecting geometry from one part into another is made *adaptive* and remains *linked* to the original component. While it is adaptive (and associated) you cannot make changes to the project geometry as it is locked so that it updates as the other component changes.

**Override** this behaviour by holding CTRL while you are projecting geometry. The geometry projects fixed in place, but not adaptive. When geometry is projected from another component (not adaptively) Inventor applies Fix constraints to ground the geometry in place.

**Adaptivity** has its pros and cons. It is a very powerful tool... when you intend to use it. Unfortunately the default behaviour makes it too easy to accidentally or inadvertently invoke adaptive.

In the **Application Options** (Assembly Tab), disable the **Cross part geometry projection settings** to reverse the default behaviour. With these options disabled you'll have to hold CTRL when projecting geometry to make it adaptive.



Quickly remove Fix Constraints by selecting the geometry, right-clicking, and selecting **Remove Fix Constraints**. This will remove all fix constraints on the selected geometry.

## Creating Geometry

### Utilizing Dynamic Input

**Dynamic Input**, originally introduced in AutoCAD, is a method of working with sketch geometry. It provides a *Heads-Up Display* (HUD) near the cursor, keeping your focus on the task at hand.

Input boxes will appear as you are creating 2D geometry showing transitional dimensional information about your geometry and it allows you to define the size of the object as you are creating it, even creating the dimension.



A key to working with Dynamic Input is the TAB key toggles between the various options, whereas ENTER will accept the sizes

Values entered become dimensions when accepted. *Dimensions* are only created on the inputs you entered a value.

Configure **Heads-Up Display (HUD)** within the **Application Options** (Sketch tab).

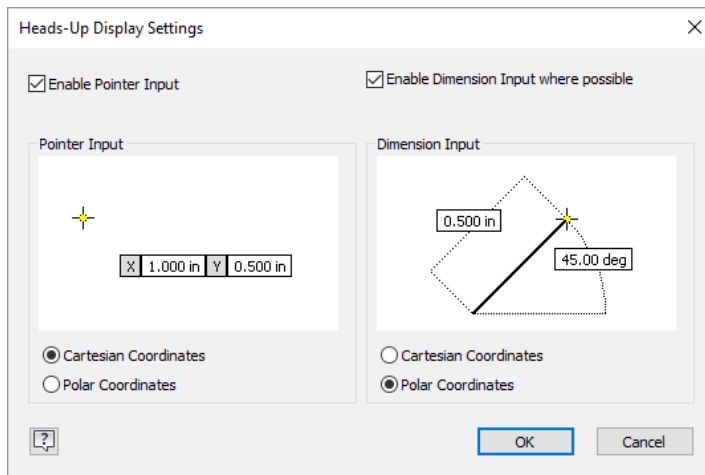


Figure 4: Heads-Up Display Settings

### Just Like AutoCAD

There are many things that Inventor mimics AutoCAD, in a good way, that provide little time savers. Anything, whether big or small, that saves time are a good thing.

- the Spacebar repeats the last command
- when drawing a line - right-click and close the line
- switch between arcs and lines in the same command

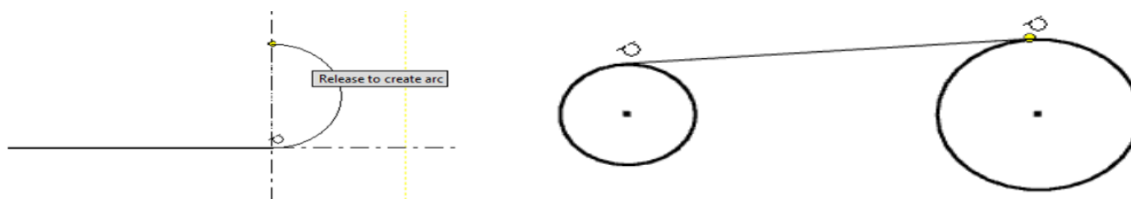


## Drawing Lines

When sketching lines use the *press-and-hold* options to generate constrained lines or switch into a tangent arc creation mode.

If you *press-hold-and-drag* off any point Inventor generates a tangent arc from that point. If you *press-hold-and-drag* away from a line or arc, Inventor creates perpendicular lines

If you *press-hold-and-drag* tangent-like from an arc or circle you create a tangent line. If you continue to *press-and-hold* as you create the line you can snap tangent at the other end as well.



## Drawing Rectangles and Slots

Inventor provides multiple methods for creating **rectangles** and **slots**. Opposed to using lines and arcs, use the appropriate option. With both rectangles and slots, select the option that provides the easiest (and best) method for locating the shape and reduces the steps to create the shape.

## Drawing Splines

Inventor provides multiple methods for generating **Splines**.

User specific equations define **Equation Curves**.

**Interpolation splines** pass through a series of points, called *fit points*. You modify the curve using handles on the points. Each fit point when selected provide handles, used to modify the curvature of the spline.

The *Control Frame* defines **Control Vertex splines**. *Control vertices* on the frame, influence the spline curves. The spline is tangent to the control frame at the start and end points. To constrain and modify the spline work with the ends of the control frames.



The **right-click menu** provides options to insert addition vertices and to convert the spline to the Interpolation type.

## Offsetting

**Offset** creates geometry by offsetting existing feature edges. Just like AutoCAD, zero-length segments are automatically removed by the offset process.



You do not need to offset the entire loop - disable **Loop Select** from the right-click menu.

## Using Split

Use **Split** to divide entities into two or more objects. When geometry is broken apart using **Split** the geometry remains constrained, meaning it is a great method to generate construction geometry or to create points to snap to.

## Adding Text to your Sketches

Add Text to any sketch and then emboss or extrude the objects to add features to your model. Use this to generate features like part numbers, dates, place of manufacture, brand names, and logos stamped, engraved, or etched into the component.

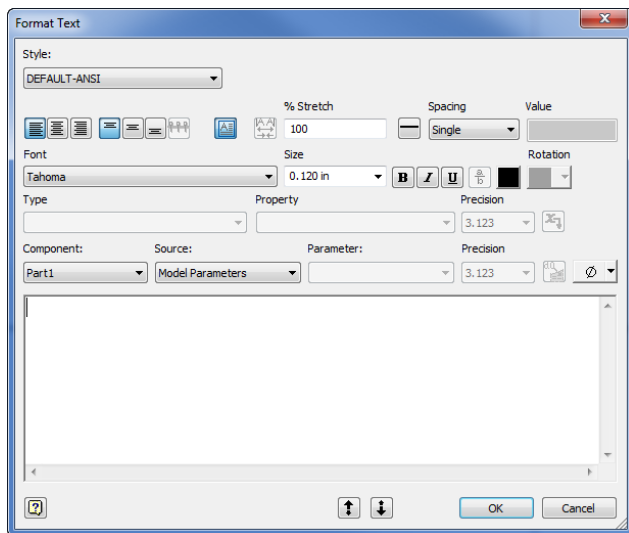


Figure 5: Text Editor

Sketched text can include model parameters meaning that as the parameters update so does the text. The location of the text can be parametrically positioned either by the insertion point or the text box can be enabled providing more fine-tuned positioning.



Figure 6: Geometry Text

**Geometry Text** functions similar but will conform the text to a selected object, for example wrapping the text to an arc.

## Using Sketches and Sketch Points to Build Hole Patterns

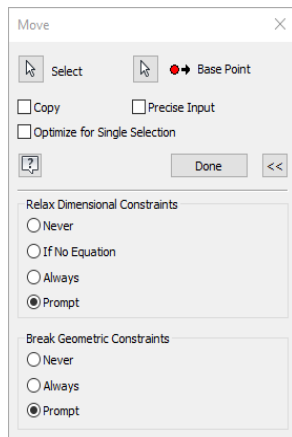
Any point in the sketch can be used as a hole center (not just hole centers). Quickly change points to hole centers by selecting the points and adjusting their format. There are many opportunities to use rectangles, polygons, and / or offset to create the hole pattern base more easily and smarter compared to patterning.

## Export Face and Export Flat Pattern

Any face or any flat pattern can be exported to DWG or DXF, avoiding the need for a drawing.



## Explaining the Options with Move, Copy, Scale, Rotate, and Stretch



Move, Copy, Scale, Rotate, and Stretch all provide an expanded portion to the dialogs. Use to define how to handle constrained geometry.

Take for example, the following sketch. The circle is fully constrained from the 0.75 dimension and a vertical constrain between the center of the circle and the midpoint of the edge of the rectangle.

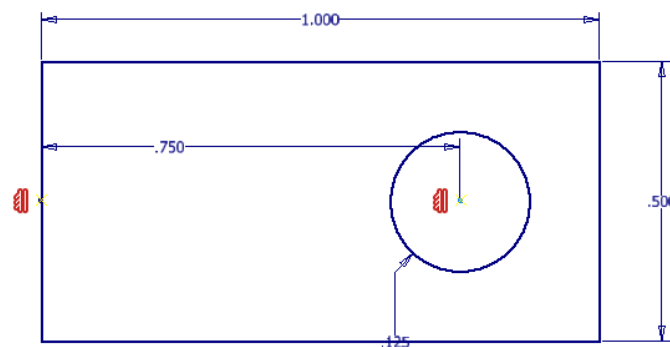
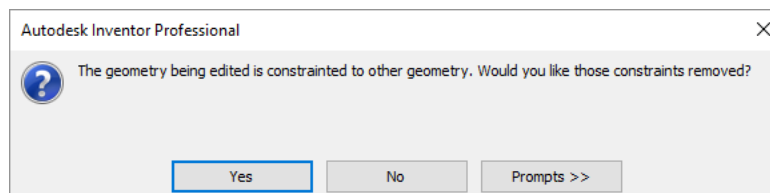


Figure 7: Example Sketch

When attempting to Move the circle, Inventor Prompts regarding the applied dimensions. If the dimension is NOT relaxed, you will be unable to move the circle left-or-right within the sketch.

Next Inventor prompts to delete the applied constraints. In this example, by not removing the Vertical constraint you cannot move the circle up-or-down within the sketch.



Use the expanded portion of each dialog to define the prompting. Select Prompt, to be prompted each time using one of these commands. Select Never, to not be prompted, and to never remove constraints nor relax dimensions. Select Always, to not be prompted, but to always remove the constraints and always relax the dimensions.

## Region Properties

**Region Properties** is a sketch-only tool. After selecting the desired areas to include select *Calculate*. Inventor provides the Area, Perimeter, Centroid, and other mechanical properties.

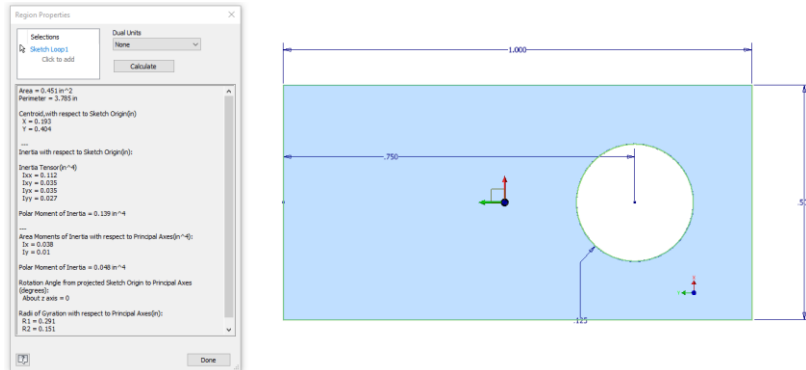


Figure 8: Region Properties

## Offsetting

Offset creates geometry by offsetting existing feature edges. As a bonus tip you do not need to select the entire loop if you disable the “Loop Select” option on the right-click menu.

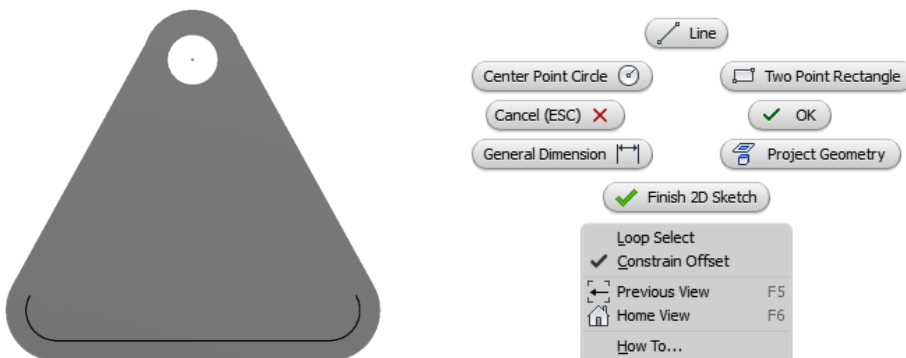


Figure 9: Using Offset

Just like AutoCAD, zero-length segments are automatically removed by the offset process.

## Sketch Options

The **Application Options** contain options specific for sketches.

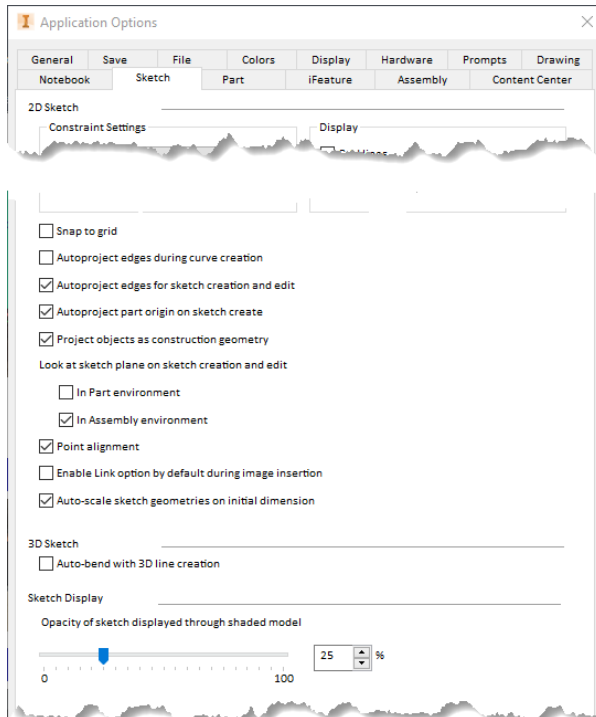


Figure 10: Sketch Options

## Projecting Options

- **Autoproject edges during curve creation** – project edges and other sketch geometry by hovering over and “rubbing” the object
- **Autoproject edges for sketch creation and edit** – auto-project the edges of the selected face onto the sketch plane as reference geometry
- **Autoproject part origin on sketch create** – automatically project the models origin point as reference geometry [THIS IS A MUST!]
- **Project objects as construction geometry** – project all geometry as construction geometry

## Look At

- **Look at sketch plane on sketch creation and edit** – when enabled, entering sketch mode automatically reorients the graphics window so that the sketch plane is parallel to the view

## Point Alignment

- **Point alignment** – enable to infer alignment between endpoints of newly created geometry. If disabled, you must pass the cursor over the points in the sketch to invoke. [NOTE: This does NOT create constraints]

## Sketch Opacity

- **Sketch Display Opacity** – manages the amount the sketch is visible through the shaded model. 0% = sketch is opaque – blocked by model.

## Using Sketches and Sketch Points to Build Hole Patterns

Any point in the sketch can be used as a hole center (not just hole centers) and you can quickly change points to hole centers by selecting the points and adjusting their format. There are many opportunities where using rectangles, polygons, and / or offset can create the basis of hole patterns more easily, smarter, or in cases where the hole pattern is non-rectangular or non-circular.

## Reusing Geometry

**Importing AutoCAD Geometry** - AutoCAD geometry can be imported into your sketches. Once it is imported, add dimensions and constraints as if the geometry originated in Inventor.

After selecting the drawing, specify the layers for import. Optionally, select the objects for import. Finalize the import by specifying the import units, if you will apply constraints, and options about converting AutoCAD Blocks to Inventor Blocks.

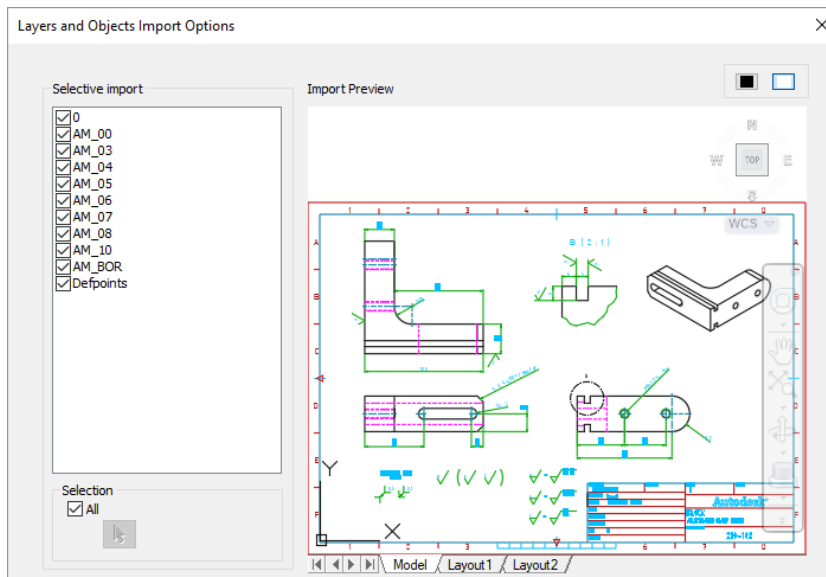


Figure 11: Import Options

**Copy & Paste** - You can copy & paste directly from AutoCAD into Inventor to bypass the wizard. Using the Windows Clipboard you can create copies of your sketch geometry into the same sketch, into the same model, or even between parts. Use the standard MS Windows functions right-click Copy or <Ctrl> + C and right-click Paste or <Ctrl> + V



Use **Auto Dimensioning** to apply the “easy” constraints like horizontal, vertical, perpendicular, and parallel

**Utilizing AnyCAD** - By utilizing **Inventor AnyCAD**, associatively import an AutoCAD drawing into your Inventor model. After selecting the drawing, specify the plane for insertion and the insertion point for the drawing.

From any sketch use **Project DWG Geometry** to project in objects from the AutoCAD drawing as reference geometry within the sketch. If the AutoCAD drawing changes, this projected geometry automatically updates.

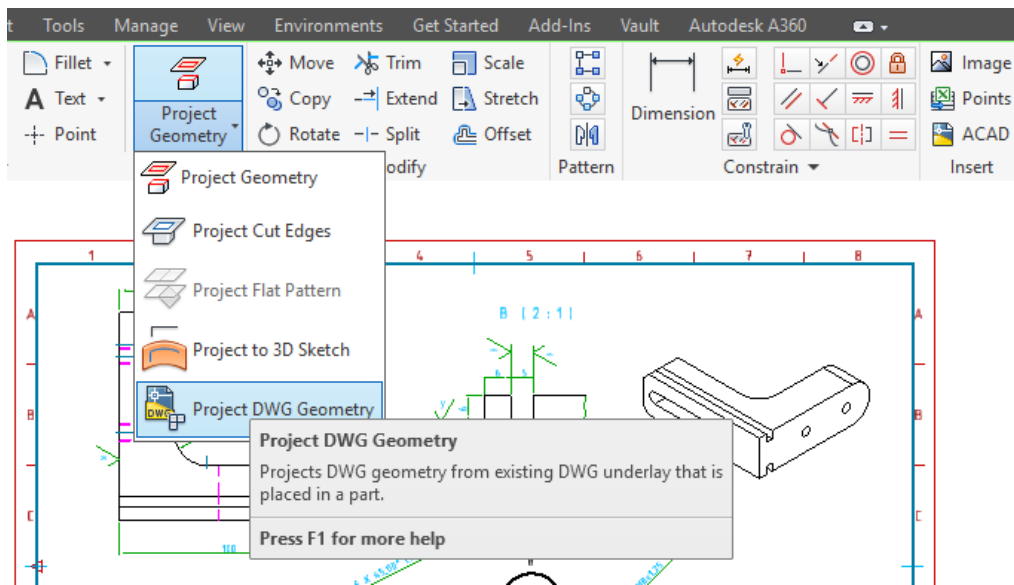


Figure 12: Project DWG Geometry

## Constraints

**Constraints** control the geometric relationships, such as parallel or tangent, between sketch entities. Sketch constraints are visible only while editing the sketch.

### Always follow the KISS rule!

**Stability before Size.** Add constraints and then add dimensions. Consider accepting the default dimensions if you are expecting “big” changes to the size of your sketch geometry. To help avoid sketch flipping issues change the smaller dimension values first then move onto the larger ones. Remember that the first dimension sets the scale of the sketch.

**Use geometric constraints as much as possible...** apply constraints first, then dimension which is more efficient and reduces the clutter. Constraints typically represent the things that are assumed on a drawing when not dimensioned

**Be Lazy and keep it simple.** There is a fine line between efficiency and laziness; Don't try to be everything to everyone with one sketch. Instead of making one sketch to be used for every feature in the model follow one of the golden rules of sketching - KISS – one sketch per feature

**Only create in the sketch what you can work with easily.** The moment sketching becomes work to fully constrained your sketch is too complicated

**Filleets are best as features** and should be avoided in the sketch, same with chamfers, mirroring and most patterns. These types of sketch elements over complicate the sketch and are better controlled as features. [Why are these tools there then? You may run into rare cases where you have to use them]

**Fully Constrain Your Sketches!** To quote Curtis Waguespack “*The importance of fully constraining your sketches in Inventor cannot be over stated.*” To make the behaviour of your Autodesk Inventor sketches more predictable, constrain your sketch to the ‘Origin’ (0,0,0).

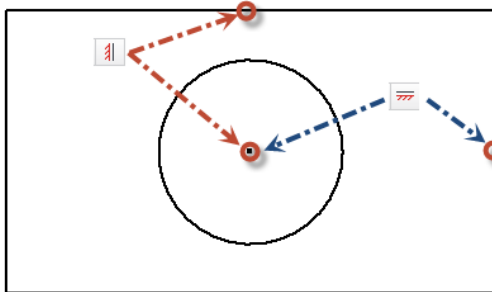


Configure Inventor to auto-project the origin point by enabling **Auto project part origin on sketch create** in the Application options



## Horizontal and Vertical can do what?

The Horizontal & Vertical constraints can be used to align points either horizontally or vertically.



The center point shown below is constrained so that it lines up horizontally with the midpoint of the vertical line and vertically with the midpoint of the horizontal line. The circle will remain constrained to these points even as the rectangle changes in size.

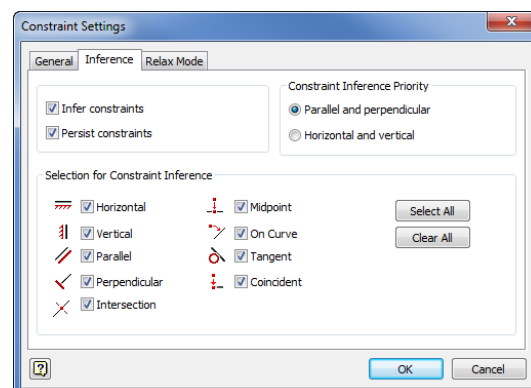
## Make Automatic (Inferred) Constraints Work for You!

**Constraints** are automatically applied as you sketch but you can make this work in your favour. This auto-constraining is referred to as constraint inference. What Inventor infers is dependent on the existing geometry, the type of object you are creating, the Constraint Options, and if Constraint Persistence is enabled.

You can adjust the geometry the auto-constraint is being inferred from by hovering (rubbing) over another object. For example if your new line is snapping parallel (without exiting the line) you can hover over a line 90-degrees to it and your new line will now snap perpendicular.

If your new geometry is inferring constraints in a fashion you do not want you can hold CTRL as you sketch and no constraints (except for coincident) will be automatically created. You can then add the constraints of your liking. Additionally you can disable Constraint Persistence on the Constrain Options dialog.

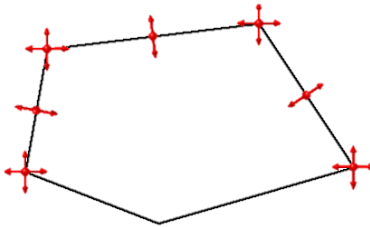
**Constraint Settings** (available on the Constrain Panel) control which constraints will be inferred during sketching



## Automatic Dimensioning – It's Not as Bad as You Think!

When you are missing constraints and just can't seem to figure out where the 1 or 2 constraints are missing the Auto-dimension feature can quickly find the missing constraint location by adding dimensions to these objects.

### Degrees of Freedom



Use **Degree of Freedom** to help find unconstrained, partially constrained or fully constrained geometry. The DOF symbols will update as you constrain your geometry.

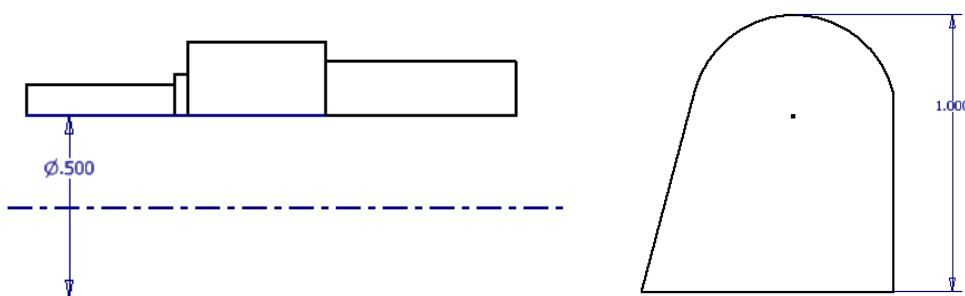
### Relax

Need to adjust the size or shape of a full constrained sketch? **Relax Mode** allows you to modify geometry and apply new constraints to geometry that is already constrained. When enabled, you can freely drag selected sketch geometry even if it is constrained, and conflicting constraints are removed when you add new dimensions.

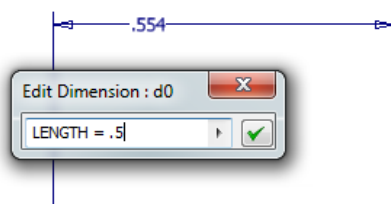
### Dimensioning Smartly

Center lines can be used to create **Revolved Sketch Dimensions**, which is a great way to dimension revolved parts like shafts.

You can create **tangent dimensions** if you know where to pick (keep moving until you see the icon change)



Save a step and build more intelligence by naming your **parameters** on the fly



## Adding Tolerances

Right-click on a dimension to access the **Dimension Properties**.

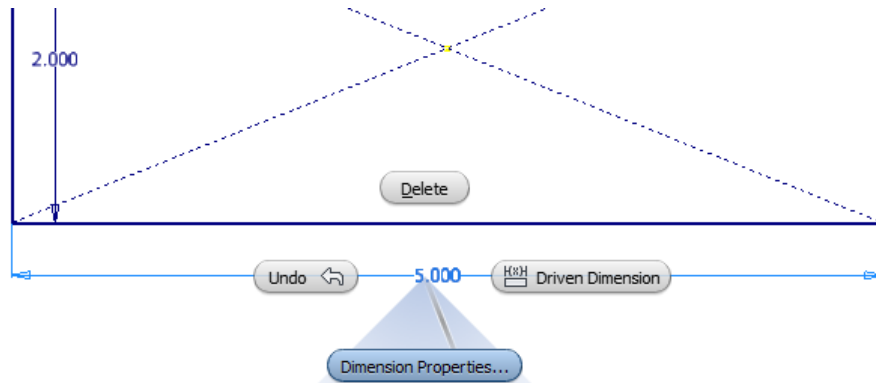


Figure 13: Dimension Properties RMB

Use the **Tolerance** section to apply tolerancing to the dimension. Use the **Evaluated Size** to set which dimensional value is evaluated within the model.

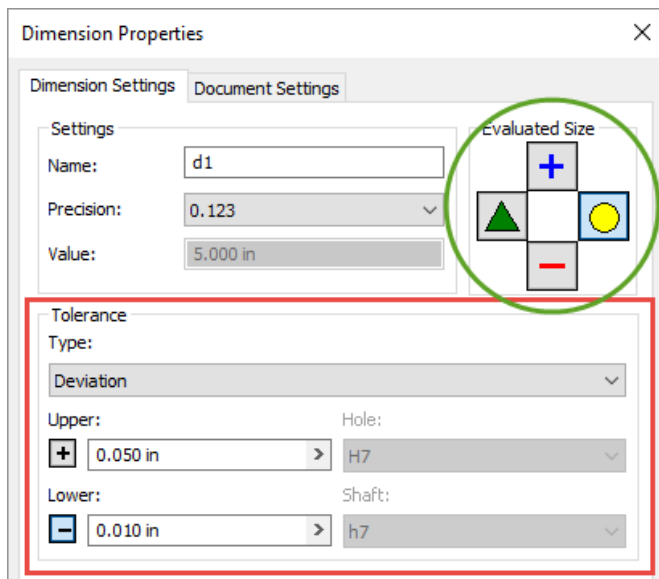


Figure 14: Dimension Properties


## Blocks

### Blocks - Not Just for AutoCAD

Similar to AutoCAD, blocks allow for the grouping of objects into a singular rigid-shape... but this isn't AutoCAD why should I use sketch blocks?

- In some situations, especially with a lot of geometry, it is easier to move and rotate portions or the entire sketch by first grouping it as a block and completing the operation on the single block entity
- Sketch blocks provide a great mechanism to test function before fit and form, even using the part sketch in an assembly
- Sketch blocks are the basis for the Layout feature to generate an assembly and parts.

Creating a block with Inventor is very similar to creating blocks in AutoCAD; You specify the name, objects, and insertion point.



AutoCAD Blocks can be easily translated into Inventor Sketch Blocks as well

The block instances will appear with the sketch and the block definitions will be listed at the top of the browser.

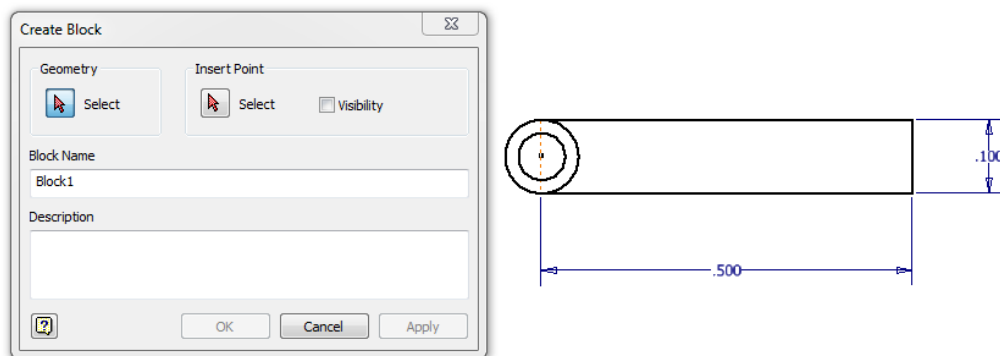


Figure 15: Create Block

Blocks can be constrained as any other 2D Sketch object using the Constraints (Horizontal, Colinear, Coincident, etc, etc) and Dimensions.

## Layouts

**Layouts** are a *top-down* design tool where the sketch is used as the “root” of the design. The 2D sketch forms the skeleton used to generate the assembly, the sketch blocks represent components you want to create. Changes to your sketch updates the components and assembly layout.

Use the layout to position components and test design feasibility.

*“The layout (with sketch blocks) are derived into part and assembly models which are associativity linked meaning changes to the layout propagates to all of the components”*

To create a Layout you

- Create a Part (.ipt) or within the Assembly use Make Layout
- Define sketch blocks representing the components you want to model
- Constrain using sketch constraints and dimensions
- Use Make Components to build the assembly and components based on the Sketch Blocks you defined

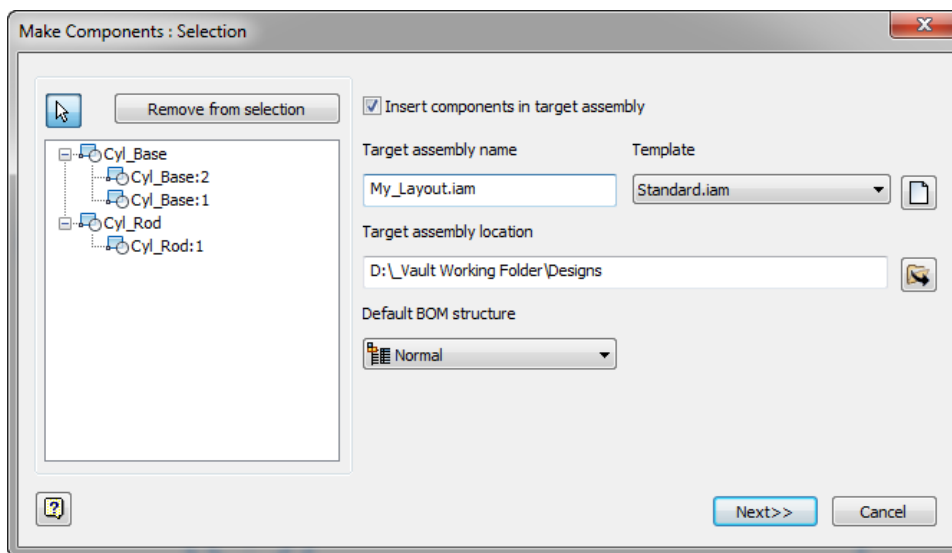


Figure 16: Make Components

The Layout component will be placed into each new component generated. The sketch constraints will be converted to assembly constraints.

## Sketches in Drawings

**Sketches** within drawings are very important. Sketches define custom borders, title blocks, and the symbols you insert. **Sketches** serve as key components for certain views like sections. Additionally, sketches are useful to add drawing elements not defined properly within the model.

### Creating & Editing Drawing Sketches

A key step to creating sketches within the drawing, is first selecting the view. This is important if you want to utilize edges from the view and for your new sketch to remain associative with the view.

	<p>Just like within the model, use <b>Project Geometry</b> to copy edges from the view for use in your sketch. The edges remain associative with the drawing view geometry.</p>
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By default, model sketches do not appear within the drawing view. Within the browser, expand the drawing view. Right-click on the model and from the model select **Get Model Sketches**.

### Creating Custom Drawing Symbols (Sketch Symbols)

Using the drawing sketch tools, a user can create their own sketch symbols. In a drawing these symbols are saved in the **Drawing Resources > Sketch Symbols** folder. They may also be saved to an external symbol library. Sketch symbols may contain geometry, text, iProperties and images.

Creating a new **Sketch Symbol** starts a new sketch. Use the rules and techniques discussed already for dimensions and constraints to produce a clean and professional looking symbol. Add and constrain text, or images as needed. Text may be linked to *iProperties*, or set as a *prompted entry* for user input.

**Insertion points** and **connection points** may be added to your symbol for more precise placement when using the symbol.

### Sketches and Drawing Views

A **Break Out view** cannot be created without an unconsumed sketch. The sketch is used to break away a segment of an existing view, in effect creating a partial section view. The sketch may be as simple as a circle, or as elaborate as the user wants to make it.

In a **Cropped view**, the rectangular border used to define the crop is an editable sketch. In the browser, expand the cropped view to reveal its sketch. Editing this sketch, allows for the creation of something other than a sharp cornered rectangular border. For example, a jagged broken edge for added effect.

In a **Section view**, the section line is an editable sketch. Use this sketch to change the path of the section line. Add constraints to precisely locate the section line. Dimensions and/or constraints may be used to ensure that the section line extension, or in the case of multiple sections, extensions are set a uniform distance from the edge of the part.