



Sketching with Autodesk Inventor

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MA1743

Class Summary

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

Learning Objectives

At the end of this class, you will be able to:

- Use sketching tips and tricks
- Effectively use constraints
- Implement Inventor dimensioning
- Manipulate geometry while sketching

About the Speaker

Mike graduated as the Most Distinguished Graduate from CAD/CAM Engineering Technology at SIAST, Kelsey Campus. He is specialist in the manufacturing industry with strong knowledge on the Autodesk mechanical products supplemented with a solid understanding of document management, hardware, networking, and other Autodesk technologies.

He is now the Technical Services Manager for [Prairie Machine & Parts](http://prairiemachine.com) and is responsible for overseeing the engineering department's technical operations and the department's strategic technical growth. Primary duties include the ongoing support of critical computer systems and programs, the interactions between the department and between other departments, providing the engineering department with effective systems and technology, and working with the VP and Engineering Managers on the development and implementation of a cohesive strategic plan for the technical growth of the department

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Sketching with Inventor

Inventor like most 3D Parametric Solid modelers supports three main types of part (component) features: Sketched, Placed, and Work. Sketch Features require a 2D sketch. The Inventor Sketch environment is used to sketch, constrain, and dimension 2D geometry

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

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

Firstly, from my 14+ years of using Inventor, I have collected a lot of tips-and-tricks regarding sketching; some from using the software and many from other users. Unfortunately I have forgotten many of the sources of these tips and tricks but there are some great places to get information on Inventor, not just sketching

The old saying is *“it’s not what you know, it’s who you know”*, but with Inventor it’s really both. I’m always on the lookout for any time saving steps that will reduce the time to get things done within Inventor or will build stronger, better models. Here are some of my favorite and some that I got a few of the tips found in this presentation:

- [Autodesk Knowledge Network](#) (AKN) – Replaces the Autodesk Wiki
- [Design and Motion](#) (Scott Moyse and John Evans)
- [TEDCF Publishing: Autodesk Inventor Tips & Tricks](#)
- [CAD Geek Speak Blog](#)
- Ascent’s [Autodesk Inventor Introduction to Solid Modeling](#)
- [Paul Munford](#) (aka the CAD Setter Out)
- [Inventor From the Trenches](#) (Curtis Waguespack)
- [Autodesk Digital Prototyping Blog](#)
- [IMAGINiT Manufacturing Solutions Blog](#) (Mark Flayler)

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Creating and Working with Sketch Planes

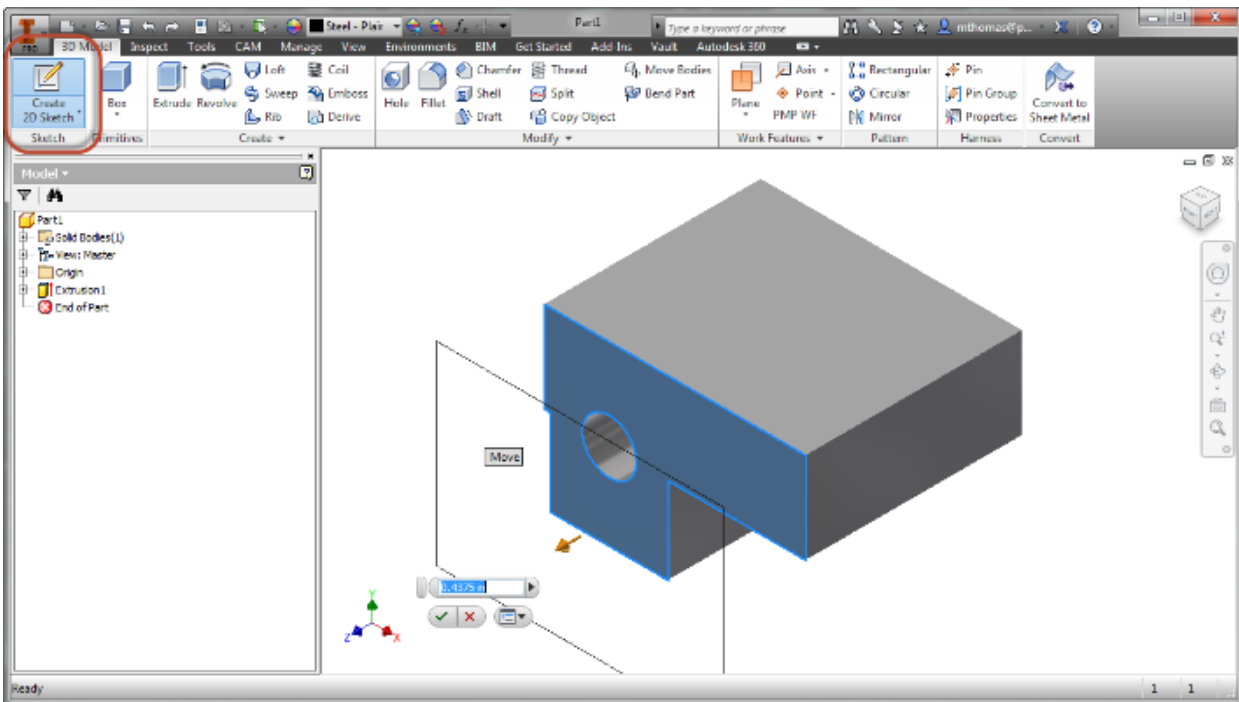
Autodesk Inventor 2014 Help Definition:

The Inventor Sketch environment consists of a sketch plane (where the sketch is located) and sketch commands. You create, edit, constrain, and dimension sketches only when the sketch environment is active. With the sketch command selected, you can specify a planar face, work plane, or sketch curve as the sketch plane.

TIP: You can create an offset Workplane while defining the sketch plane

COMMANDS: Create 2D Sketch

You can generate an offset workplane with a sketch on it in one step by dragging off the desired face with the create sketch command active



TIP: Sketches can be used to define Work Features

COMMANDS: Create 2D Sketch, Work Plane, Work Axis

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

For an example look at Ben Curtin's example [Inventor Holes at a Compound Angle](#) on the Tata CAD Geek Speak blog

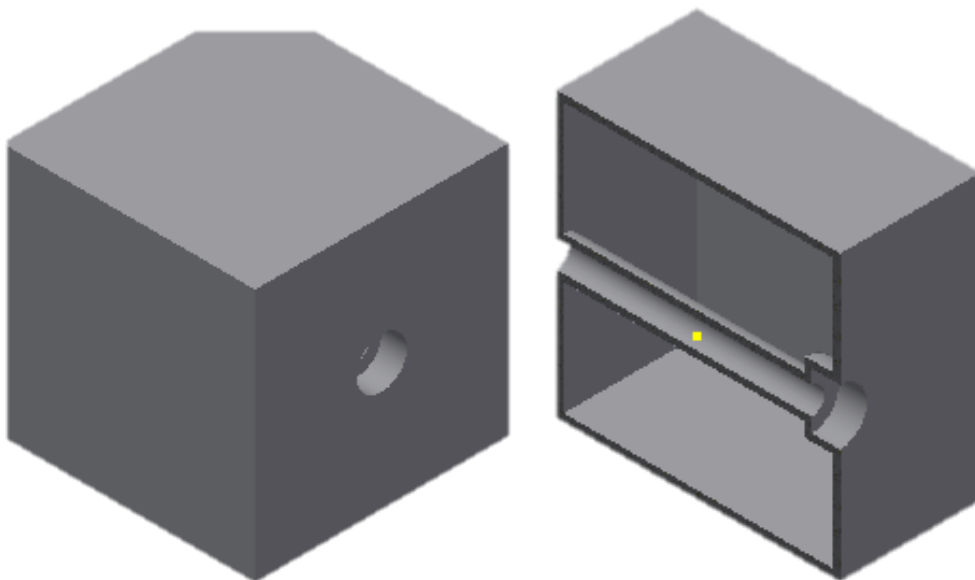
TIP: Slice Graphics

Autodesk Inventor Help Definition:

Sometimes geometry obscures your sketch plane, or other components in a part model hide it. When the sketch tool is active, use the context menu option Slice Graphics to slice away temporarily the portion of the model that obscures the plane.

When the model is in the way on your sketching Slice Graphics removes everything between you and the sketch plane.

You can initiate Slice Graphics when the editing a sketch via the right-click menu or by using F7. If by chance the model is sliced in the wrong direction, turn off the slice graphics, rotate the model, and try slice graphics again

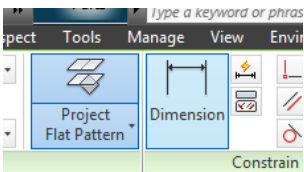


TIP: Project Cut Edges**COMMANDS: 2D Sketch**

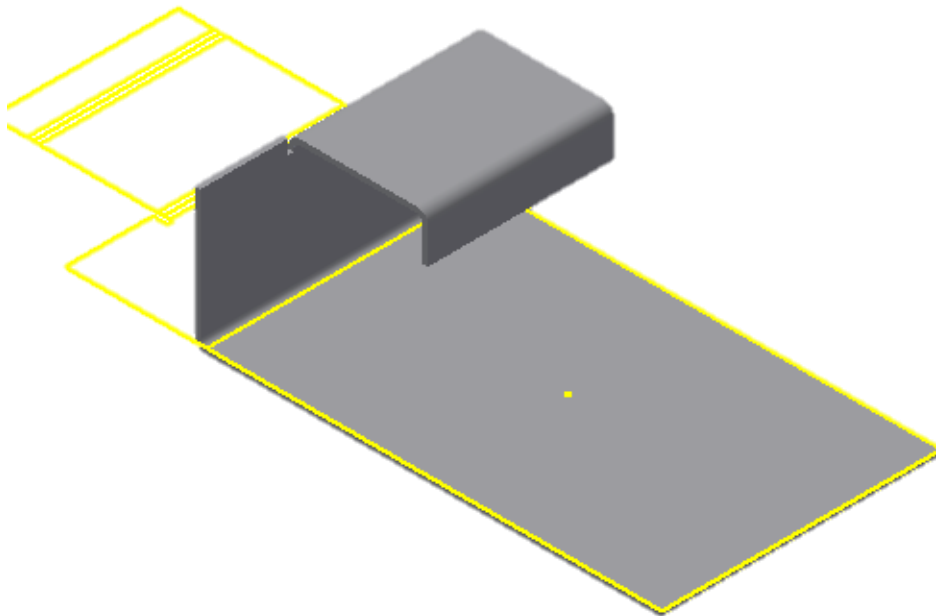
Use Project Cut Edges to associatively project edges of the model that intersect with the sketching plane. Think of it as the edges of a section view and your sketch is the section line

TIP: Project Flat Pattern

Autodesk Inventor Help Definition: *"Unfolds a disjointed face or faces into the sketch plane."*



Project Flat Pattern is great for the situations where you don't need the overhead and advanced options of the Sheet Metal Unfold / Refold features, just wanting to reference existing geometry edges



Creating Geometry

Autodesk Inventor 2014 Help Definition:

A Sketch begins when you sketch lines, arcs, and other geometry on a sketch plane . Constraints are automatically applied as you sketch. The sketch becomes a profile that is the basis for sketched features such as extrusions, revolutions, lofts, coils, or paths for sweep features. In drawings, sketch geometry creates symbols, title blocks, and borders.

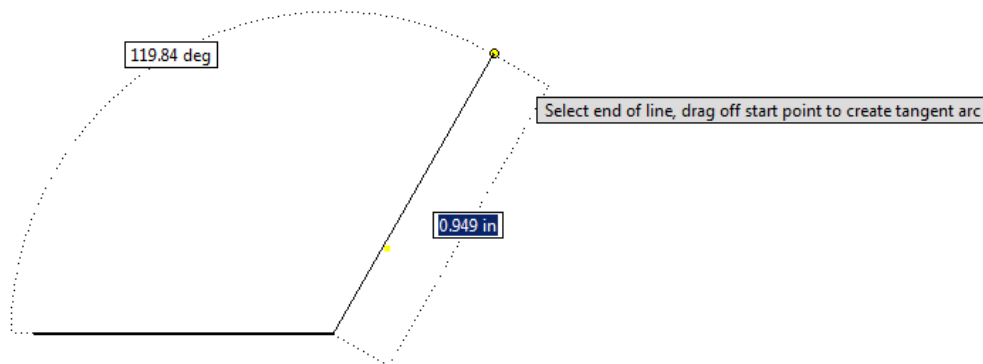
Sketches often include construction and reference geometry. Construction geometry is not part of the profile or path, but infers constraints. Reference geometry is usually part of a feature, such as an edge, or silhouette. You project it from the sketch of the feature to a new sketch.

TIP: Use Dynamic Input

COMMANDS: Any 2D creation command in the Sketch Environment

Dynamic Input is a way of working with sketch geometry that was originally introduced in AutoCAD but has been available in the sketch environment for the past few releases of Inventor. It “*provides a Heads-Up Display (HUD) interface near the cursor to help you keep your focus in the sketching area.*”

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.



Key aspect of 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 [disable Persistent Dimensions if you don't want dimensions created]. Dimensions are only created on the inputs you entered a value

TIP: Some things work like AutoCAD

COMMANDS: Many 2D creation command in the Sketch Environment

There are many things that Inventor mimics AutoCAD, in a good way, that provide little time savers. Anything, where big or small, that saves time are a good thing as even the little time savers over a day or week can add up.

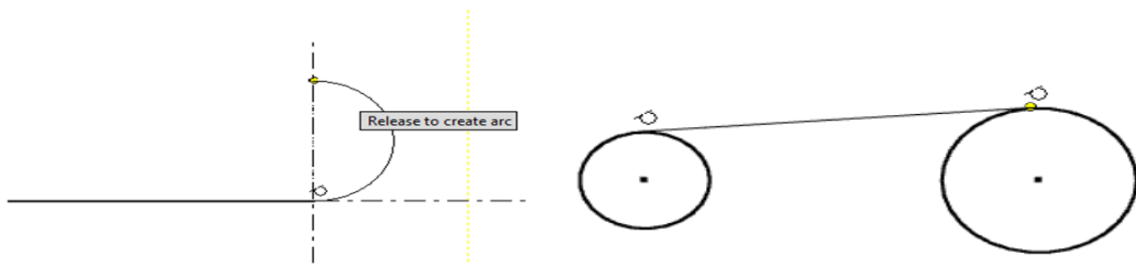
- **Just like AutoCAD the Spacebar repeats the last command**
- **Just like AutoCAD when drawing a line you can right-click and close the line**
- **Similar to how with AutoCAD polylines you can switch between arcs and lines in the same command you can within Inventor drag in the line command to create an arc**

TIP: Options when Sketching Lines

COMMANDS: Line

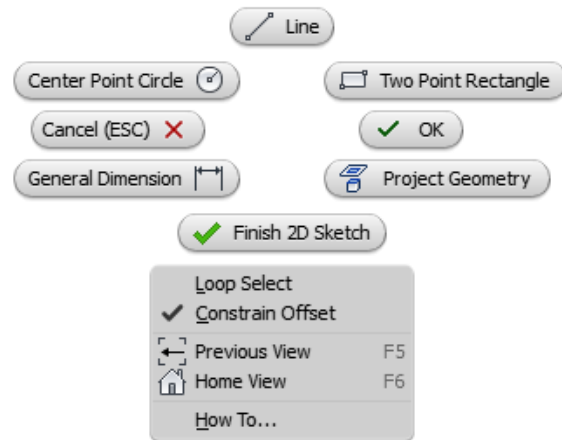
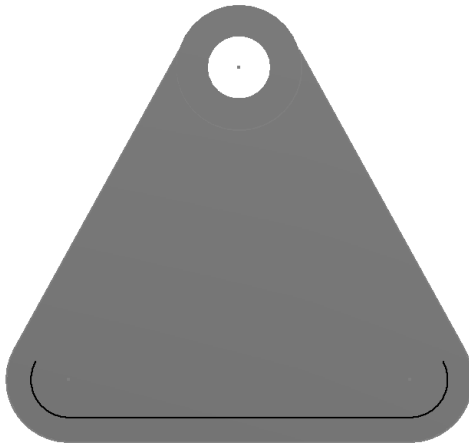
When sketching lines there are some press-and-hold options to generate constrained lines or temporarily switch into a tangent arc creation mode.

- **If you press-and-hold off the end of any point Inventor will generate a tangent arc from that point**
- **If you press-and-hold and drag away from a line or arc Inventor will create a perpendicular line**
- **If you press-and-hold and drag tangent-like from an arc or circle you will 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.**



TIP: Use Offset**COMMANDS: Offset**

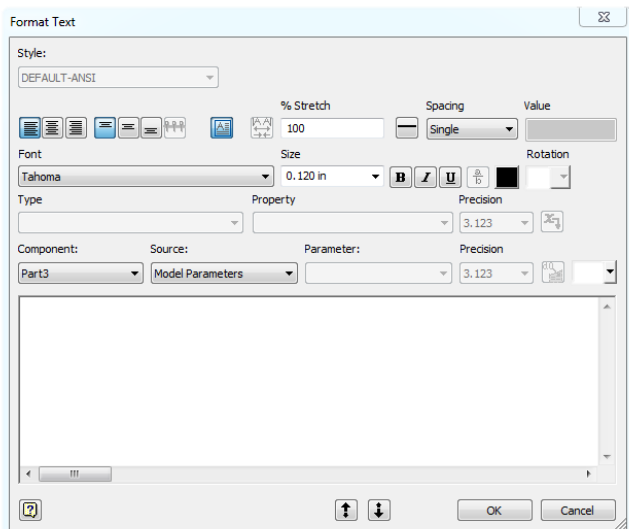
Offset creates geometry by offsetting existing feature edges. As a bonus tip you do not need to select the entire loop if you disable the “Loop Select” option on the right-click menu



TIP: You can create Text in the Sketch Environment

COMMANDS: Text, Geometry Text

Text can be added to any sketch and then embossed or extruded to add features to your model. This can be used to generate things like part numbers, dates, place of manufacture, brand names, and logos stamped, engraved, or etched into the component



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

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



Reusing Geometry

Why redo when you can reuse?

TIP: Import AutoCAD Geometry

COMMANDS: AutoCAD Geometry Import

Why redo when you can reuse? AutoCAD geometry can be imported into your sketches and once its imported you can add dimensions and constraints as if the geometry originated in Inventor.

Bonus TIP: You can copy & paste to bypass the wizard

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

TIP: Copy and Paste using the Windows Clipboard

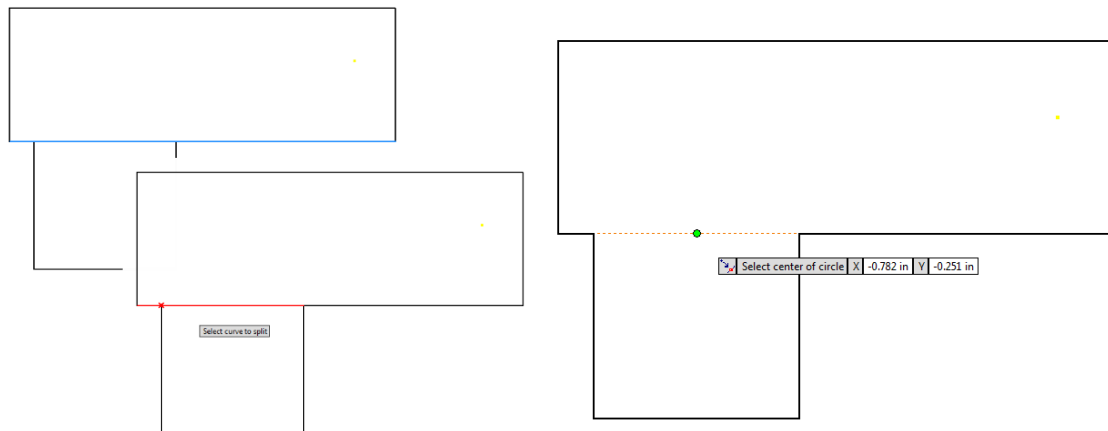
COMMANDS: Copy, Paste

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

TIP: Use Split to adjust your geometry

Autodesk Inventor Help Definition: *"Split curve entities into two or more sections"*

When geometry is broken apart using the Split feature the geometry remains constrained meaning it is a great method to generate construction geometry or to create points to snap to



TIP: Make Project Geometry Work for You

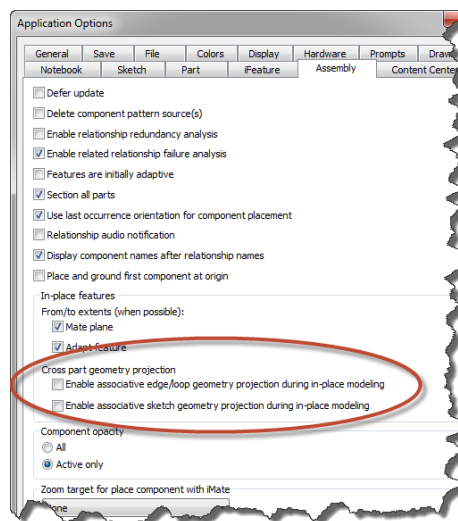
Autodesk Inventor Help Definition: *"Split curve entities into two or more sections"*

By default when you project geometry from one part into another it is made adaptive and remains linked to the other component. While it is adaptive and associated with the other component you cannot make changes to the project geometry because it will update as the other component changes.

You can override this behaviour by holding CTRL while you are projecting geometry, the geometry will be projected fixed in place opposed to adaptive

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

You can disable the Cross part geometry projection options in the Application Options (Assembly Tab) to flip the default behaviour. With these options disable you'd have to hold CTRL when projecting geometry to make it adaptive.



If you haven't yet I highly-recommend disabling the two Cross part geometry project options so that you do not accidentally create adaptive, cross-part associative geometry.

When geometry is projected from another component but not made adaptive Inventor applied Fix constraints to ground the component in place. You could Show Constraints and delete all the fix constraints, but there is a faster way. Select all the geometry you want to be free, that you want to dimension or constrain, right-click, and select Remove Fix Constraints. This will remove all fix constraints on the selected geometry.

Constraints & Dimensions

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

TIP: Always follow the KISS rule!

COMMANDS: Keep It Simple Stupid!

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

Fillets 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]

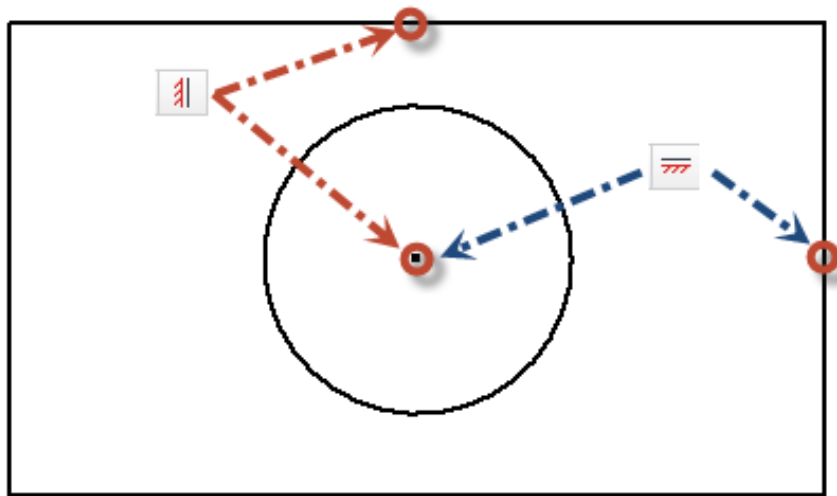
Fully Constrain Your Sketches! To quote Curtis Waguespack “*The importance of fully constraining your sketches in Inventor cannot be over stated.*” To make the behaviour of your Autodesk Inventor sketches more predictable, constrain your sketch to the ‘Origin’ (0,0,0).

BONUS TIP: You can have Inventor auto-project the origin point by enabling ‘Auto project part origin on sketch create’. [Tools>Application options>Sketch tab]. Be careful with fix constraints, you should never need more than one

TIP: Consider using Horizontal and Vertical for alignment

COMMANDS: Horizontal Constraint, Vertical Constraint

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

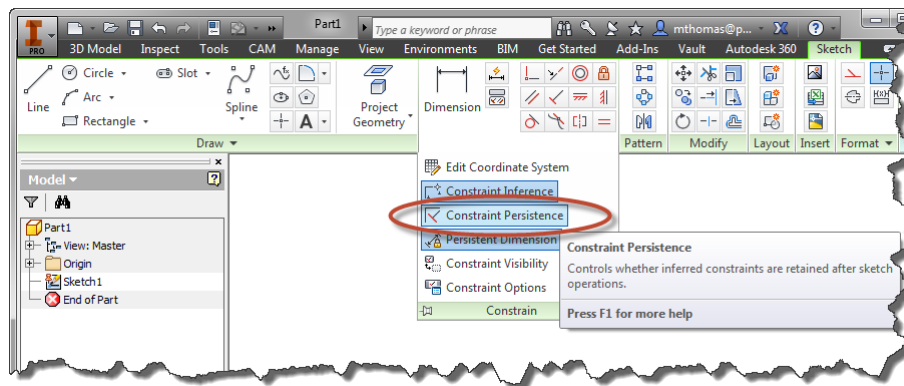


TIP: Make Automatic (Inferred) Constraints work for you

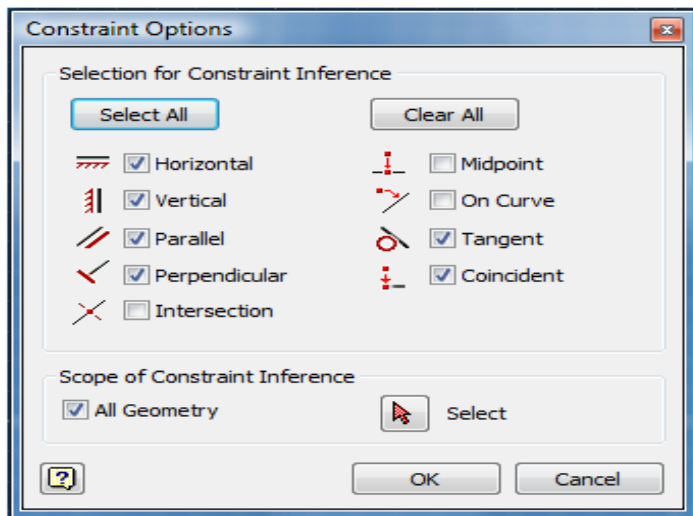
COMMANDS: Constraint Options

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 dependant 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 Panel flyout.

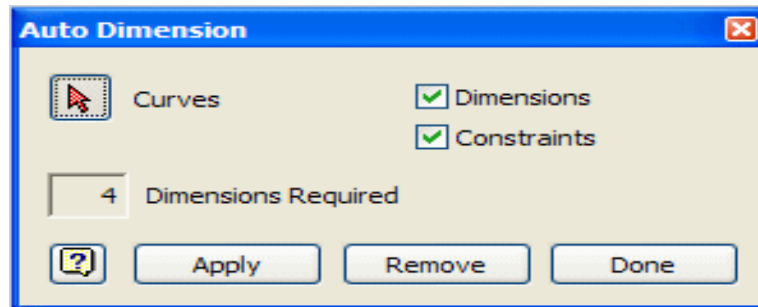


Constraint Options (available on the Constrain Flyout) control which constraints will be inferred during sketching

TIP: Auto Dimension can be a quick way to determine where you are missing constraints

COMMANDS: Auto Dimensions

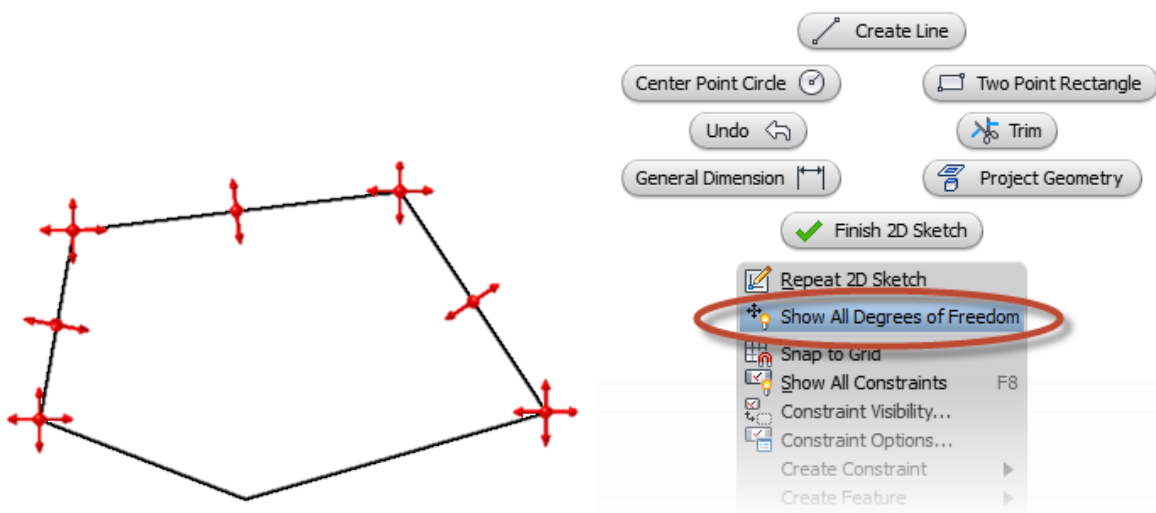
When you are missing constraints and just can't seem to figure out where the 1 or 2 constraints are missing the Auto-dimension feature can quickly find the missing constraint location by adding dimensions to these objects.



TIP: Use Degree of Freedom to find under constrained geometry

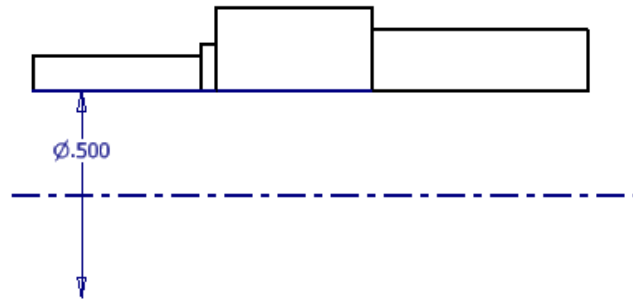
COMMANDS: Degree of Freedoms (Sketch Environment)

Use Degree of Freedom to help find unconstrained, partially constrained or fully constrained geometry. Easiest method to access is via the right-click menu "*Show All Degrees of Freedom*". The DOF symbols will update as you constrain your geometry

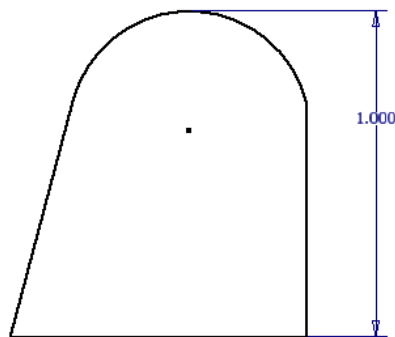


TIP: Dimension Smartly**COMMANDS: Dimension**

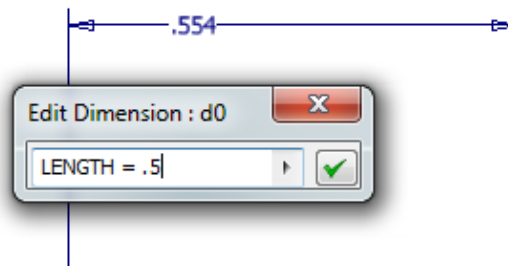
Center lines can be used to create Revolved Sketch Dimensions, which is a great way to dimension revolved parts like Shafts.



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



Other Tips, Tricks, and Suggestions

Other time savers and methods of building robust-smart sketches

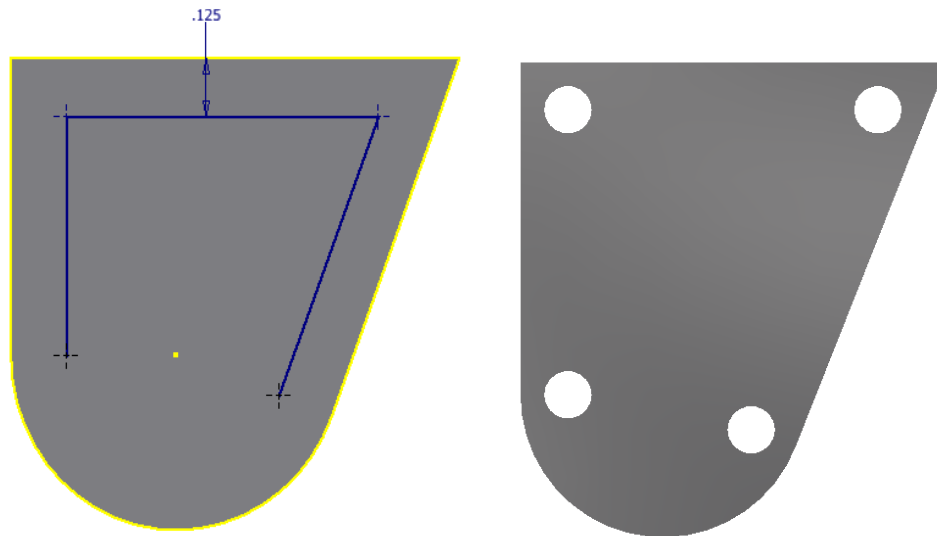
TIP: Use Sketches and Sketch Points to build hole patterns

COMMANDS: Dimension

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.



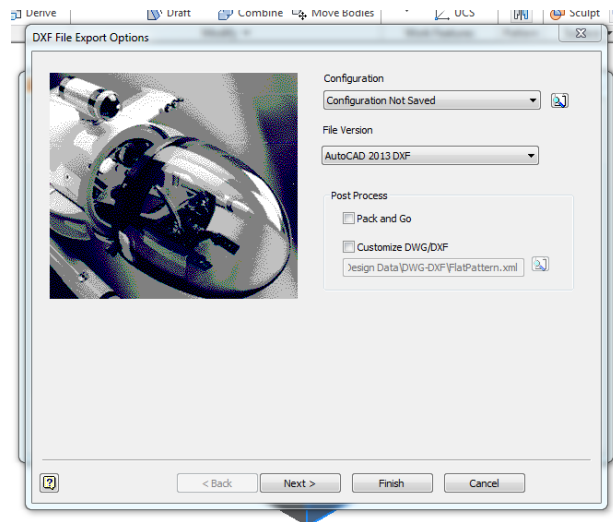
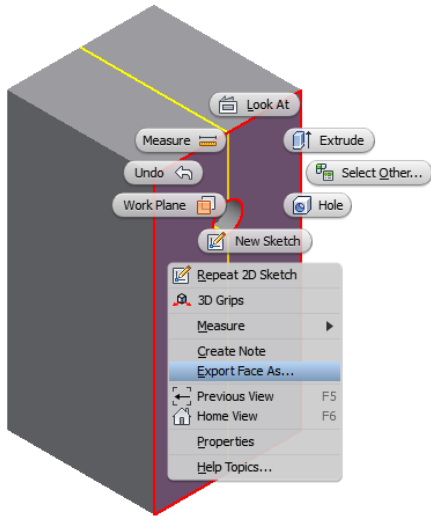
There are many opportunities where using rectangles, polygons, and / or offset can create the basis of hole patterns more easily, smarter, or in cases where the hole pattern is non-rectangular or non-circular.



TIP: Any face in the model or any flat pattern can be exported to DXF

COMMANDS: Export Face, Export Flat Pattern

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



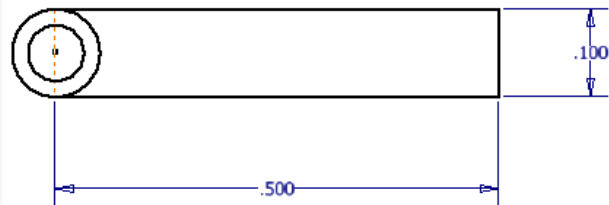
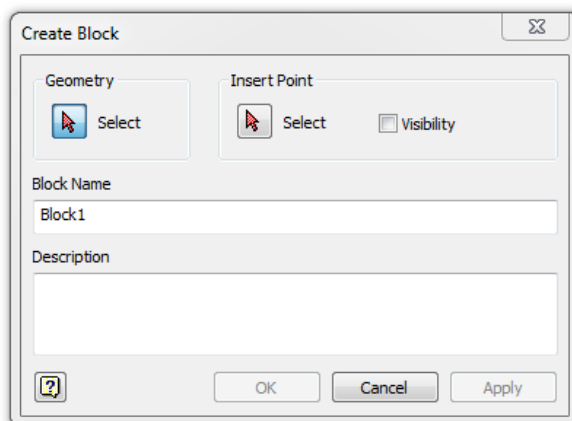
TIP: Use Sketch Blocks**COMMANDS: Create Block**

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?

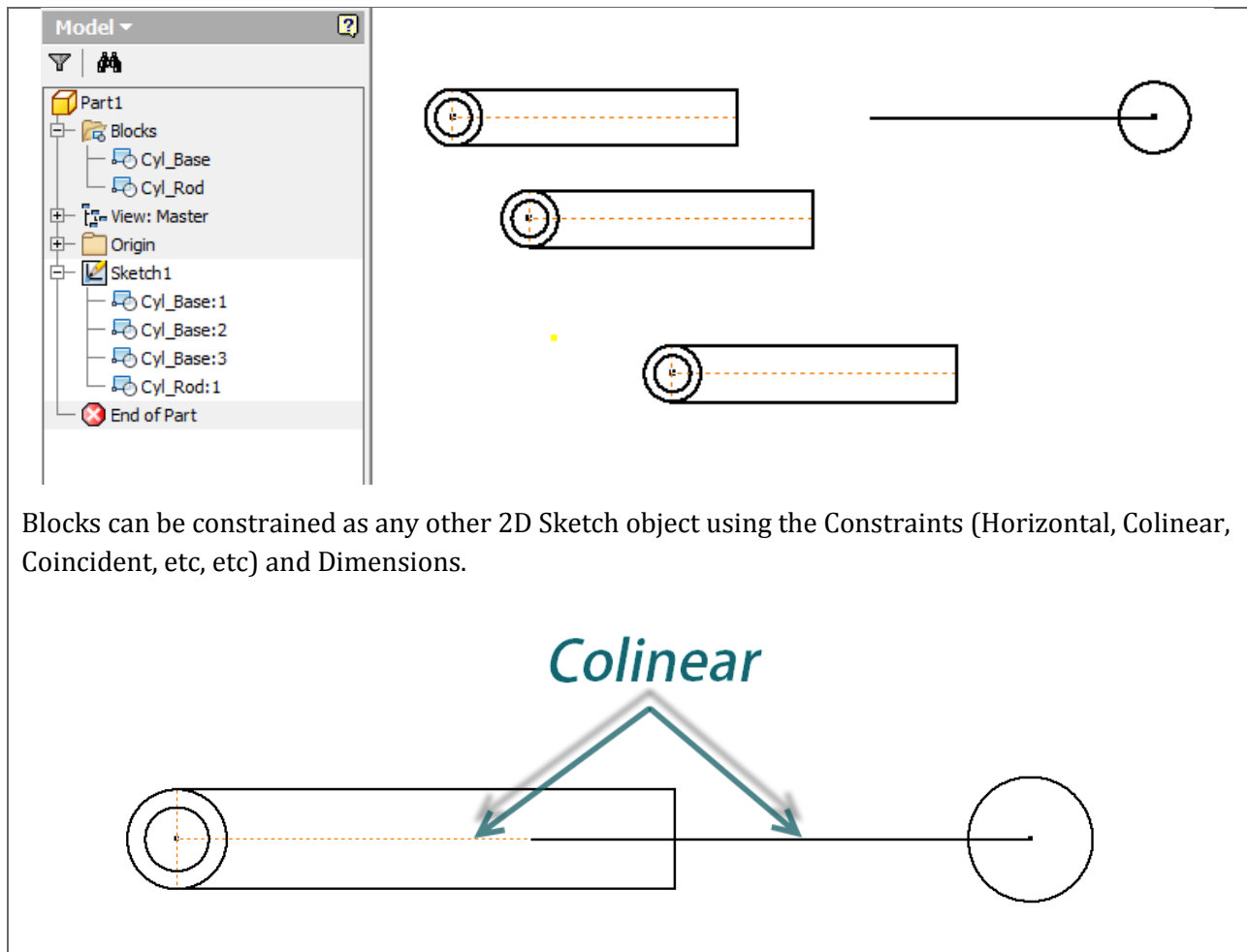
- In some situations, especially with a lot of geometry, it is easier to move and rotate portions or the entire sketch by first grouping it as a block and completing the operation on the single block entity
- Sketch blocks provide a great mechanism to test function before fit and form, even using the 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. [The insertion point is the point where the block is inserted by when placed into your sketch]

NOTE: 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.



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

TIP: Use Layouts to build your assemblies (Skeleton Modeling)

COMMANDS: Layouts

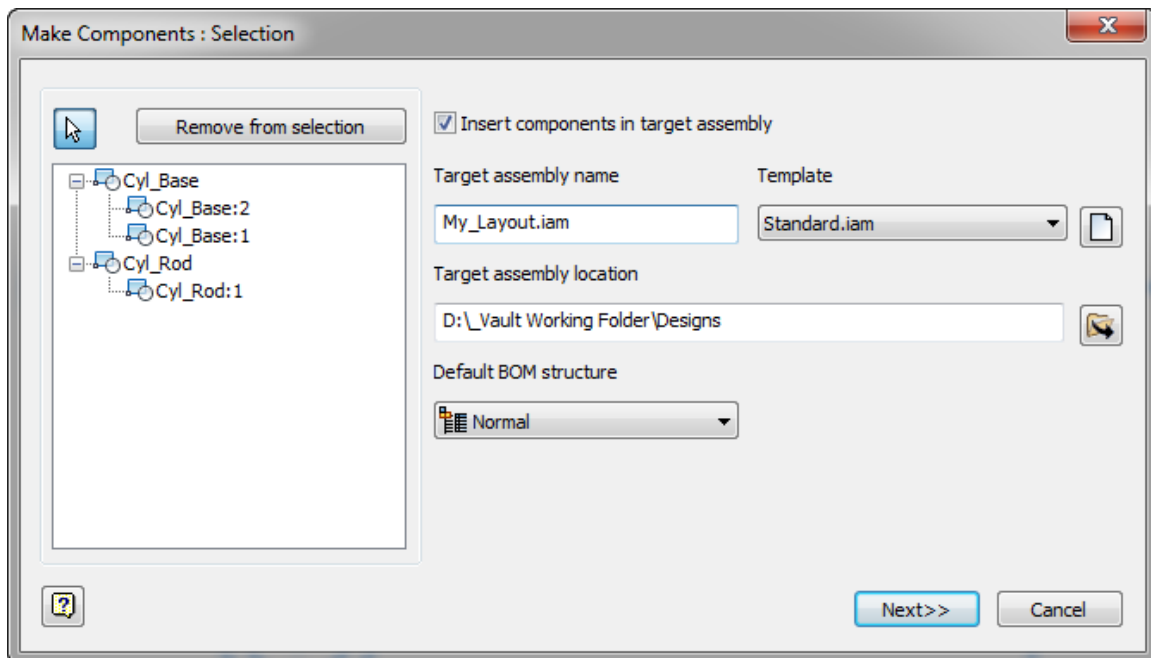
Layouts are a Top-down design tool where the sketch is used as the “root” of the design. The 2D sketch forms the skeleton used to generate the assembly, the sketch blocks represent components you want to create. Changes to your sketch updates the components and assembly layout.

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



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