The Power of Constraints Within Dynamic Blocks

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AU318-1 Learn how to make your block libraries more powerful by using Constraints within Dynamic Blocks. Through a series of real-world examples, learn how to efficiently apply geometric and dimensional constraints to manage the geometry inside a dynamic block, how to expose custom properties and grips to manipulate the block, and how to represent families of parts using dynamic blocks with Block Properties Tables. This class will benefit anyone who uses blocks in AutoCAD® or maintains block libraries in any setting. Intended for advanced users from all industries.

About the Speaker:

John has worked for 16 years in the AutoCAD® development team at Autodesk. He was one of the lead engineers on the dynamic blocks feature in AutoCAD 2005, but has worked on many AutoCAD features, including the introduction of parametric constraints in AutoCAD 2010. He started using AutoCAD with Release 11 in 1991 while he was working for an Autodesk developer before joining Autodesk. John holds a Master of Science degree in Structural Mechanics from the University of California at Los Angeles.

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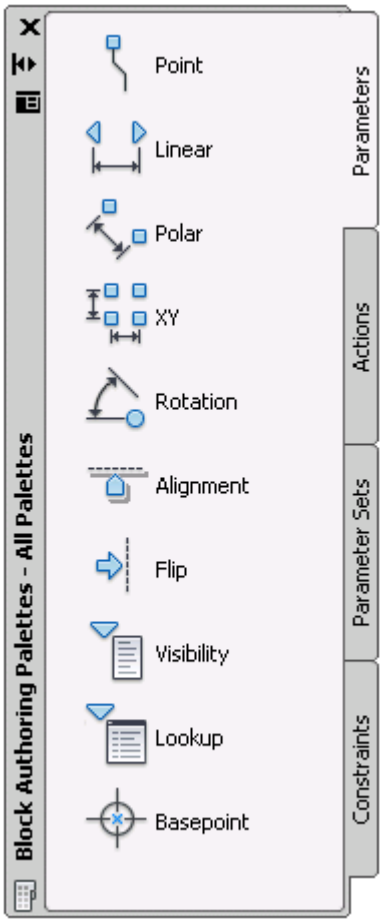
Dynamic Blocks Review: Custom Properties and Custom Grips through Block Actions and Block Action Parameters

AutoCAD first introduced blocks more than 20 years ago, while Dynamic Blocks first appeared in AutoCAD Release 2005. Dynamic blocks offer the same advantages for managing drawing content that older blocks offered:

- You define the content of a block once. The definition contains entities that are common to all references of the block. Block definitions can contain block attributes that are text appear as text whose values may be different for different references to the block.
- You insert multiple references to the definition into your drawing, each with its own location, rotation, and scale.
- When you update the definition, all of the references update automatically *except for attribute changes.*
- You use ATTSYNC to update existing block references with changes to block attributes in the definition.
- You extract information about the blocks, such as the number of blocks in the drawing or attribute values for blocks to tables in the drawing or to external files using AutoCAD's *Data Extraction* feature.

Dynamic Blocks extend the capabilities of traditional blocks by providing the ability to define custom grips and properties for your blocks which affect the geometry for the block. With dynamic blocks you can use one block definition that has a different visual appearance, and different properties, for each instance of the block in your drawing.



You create dynamic blocks by combining *Block Actions* and *Block Action Parameters* within the block definition.

- You define custom properties and grips for your block by adding one or more block action parameters. Some parameters behave like associative dimensions: they define a property that measures a distance, coordinate, or angle, and provide grips for changing the property value. Other parameters control entity visibility or align the block to other geometry in your drawing.
- You define how a change to a custom property (or grip) affects the geometry for the block reference by combining an Action Parameter with one or more *Actions*. Actions behave like commands you use to modify entities in AutoCAD and have similar names: MOVE, ROTATE, STRETCH, SCALE, MIRROR (called "Flip" in Dynamic Blocks), and others.
- You define relationships between custom properties using Lookup Parameters and Lookup Actions. The table defines the allowed combination of property values supported by the block.

Block Libraries

The best way to re-use your investment in blocks is by organizing them into *block libraries*. A block library is nothing more than a drawing file containing a collection of block definitions. Depending on the number of blocks in your library you may organize them into drawings of related blocks (e.g. one drawing for valve

blocks, another drawing for instrument blocks, another drawing for vessel blocks, etc.). By organizing your blocks into libraries containing many definitions you can take advantage of the tools AutoCAD provides for quickly accessing your block library, including Tool Palettes and Design Center. It is also easier to share and manage your blocks when organized into libraries as opposed to placing them into individual drawing files.

Constraints and Dynamic Blocks

With AutoCAD 2010 you can create *Parametric Drawings* by defining geometric and dimensional relationships for the geometry in your drawings and blocks. For example you can define geometric relationships keep two lines parallel, or perpendicular, or will keep two circles concentric. You can define dimensional relationships that maintain the length of a line, or that maintain the angle between two lines.

You can also define formulas that compute one relationship from another. For example you can require the length of one line to be twice the length of another line, or you can require a rectangle to have a constant area by defining relationships between the lengths of the sides.

Constraints allow you to create drawings that are easier to maintain, harder to “break”, and contain relationships that embody the real intent of the drawing. When you combine constraints with Dynamic Blocks you add another level of intelligence to your drawings and really unleash the power of the computer to create intelligent documents.

AutoCAD 2010 provides a number of Dynamic Blocks features to leverage the power of constraints. These include:

- *Geometric and Dimensional Constraints*. You can define simple geometric relationships for geometry in your blocks that AutoCAD maintains as you modify the block.
- *Constraint Parameters*. Similar to Action Parameters, Constraint Parameters expose custom grips and properties of your blocks, but use dimensional constraints to manage the geometry in your block instead of block actions. Constraint Parameters can depend on each other through expressions you define for the parameter variable.
- *Block Properties Table*. A more powerful version of lookup tables, the Block Properties Table defines the allowed combinations of the custom properties in your dynamic block. You can include properties from action parameters, constraint parameters, user variables, and block attributes.
- *Construction Geometry*. AutoCAD allows you to convert regular geometry in your block definition into construction geometry. AutoCAD hides construction geometry when you insert the block but allows you to see and interact with it in the block editor. As the name implies you use construction geometry to construct the location of other geometry in your block, and becomes particularly useful when used with constraints.
- *Identifying Constrained Geometry*. Understanding how well the geometry is constrained in your block is important to successfully using constraints in dynamic blocks. AutoCAD provides a way to visually identify how well each entity in your block is constrained.

Block Action Parameters and Constraints in Dynamic Blocks

Whether you use traditional action parameters or constraints (or some combination of the two) in your blocks depends on how you want to express the way AutoCAD manages the geometry in the block. With both approaches you express:

- *Modifications* that should occur to the geometry in the block, for example two lines that should move when a property changes.
- *Relationships* that should be maintained within the block, for example two lines that should remain parallel.

The key difference between constraints and actions is how you express (or think of) the modifications and relationships.

- Block action parameters *directly* express *modifications* that occur when a custom grip or property changes. You *indirectly* express relationships for geometry in the block by the way the modifications affect the geometry. For example, if two lines are initially parallel they will remain parallel if the move action you assign to one of the lines moves it along an axis parallel to the lines.
- Constraints directly express *relationships between* the geometry in your block. The *modifications* to the geometry within the block are obtained indirectly when AutoCAD applies the constraint to maintain the relationships. For example, if you constrain two lines to be parallel and add an action which rotates one of the lines, AutoCAD might move, rotate, or stretch one or both lines to maintain the parallel constraint.

In many you will find that you can achieve a particular behavior equally well using only block action parameters *or* only constraints. Which to use is a matter of personal preference. Keep in mind the following when deciding what approach to use for a particular block definition

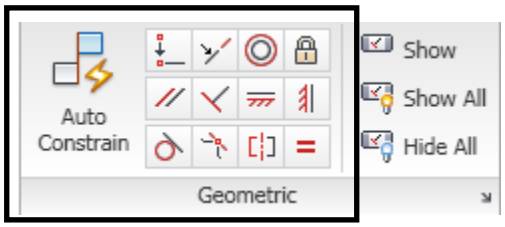
- You normally *fully constrain* geometry in dynamic blocks that containing *only* constraints.
- You normally *partially constrain* geometry in dynamic blocks containing *both* constraints and block actions.
- Dynamic blocks with constraints lead to larger drawings than dynamic blocks without constraints.
- Dynamic blocks with constraints behave as anonymous blocks (no custom grips or properties) in all releases prior to AutoCAD 2010. Dynamic blocks with block action parameters behave as anonymous blocks in releases prior to AutoCAD 2005.
- Performance is worse for blocks containing constraints.
- Easy to understand blocks are easy to maintain and update. There is value in using the approach that naturally “fits” with the way the block behaves.

2D Parametric Constraints Overview

2D Parametric constraints in AutoCAD allow you to define persistent relationships between geometry in your drawings. You apply geometric or dimensional constraints to existing geometry to establish the persistent relationships. Relationships can only be created between co-planar 2D entities, but the entities can lie in any plane and are not limited to the world X-Y plane.

Once established AutoCAD maintains the constraints until you remove them. You can use normal commands to modify constrained geometry (e.g. MOVE, ROTATE, grip-editing, etc.) and AutoCAD maintains the relationships between the geometry while performing the operation. In some cases this means that the operation has no effect, for example you might try to move a line that is fixed at both ends: AutoCAD will not move the line. You might try to ROTATE a line that has a horizontal constraint: depending on the other constraints on the line it might move or do nothing, in any case it remains horizontal.

Geometric Constraints



Geometric constraints define persistent relationships based on basic geometric properties of the entity or entities you apply them to. Geometric constraints can be:

- Coincident
- Co-linear
- Tangent
- Perpendicular
- Parallel
- Horizontal (relative to the current UCS X axis)
- Vertical (relative to the current UCS Y axis)
- Concentric
- Equal
- Symmetric
- Smooth
- Fixed

Applying geometric constraints

You use the GEOMCONTRAINT or AUTOCONSTRAIN commands to create geometric constraints.

The AUTOCONSTRAIN command examines entities you select and attempts to automatically constrain the geometry based on its current position. The AUTOCONSTRAIN command never modifies the geometry when applying geometric constraints.

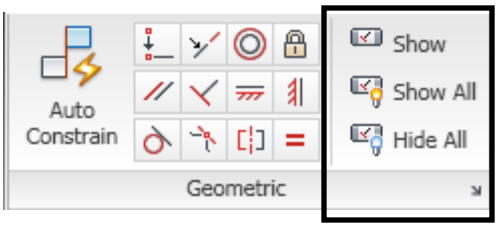
For example it might apply horizontal constraints to lines which are parallel to the current X axis; it might apply parallel constraints between parallel lines; it might apply coincident constraints between lines which share end points. You can customize the type of geometric constraints the command applies using the CONSTRAINTSETTINGS dialog.

Pitfall: AUTOCONSTRAIN helps you to save time by applying constraints in simple and obvious situations, but avoid the temptation to use it everywhere. Always verify the constraints created but AUTOCONSTRAIN and do not be tempted to blindly apply it to your entire drawing.

The GEOMCONSTRAINT command creates geometric constraints one at a time. It prompts for the constraint type to create and for the geometry to apply the constraint to. When the geometry does not satisfy the newly applied constraint AutoCAD modifies the geometry satisfy the constraint. When applying constraints between two entities AutoCAD modifies the second entity selected where possible.

Managing geometric constraints using constraint bars

AutoCAD's *Constraint Bars* provide a heads-up interface to help you manage geometric constraints in your drawings. You use the CONSTRAINTBAR command to show or hide control bars for geometry in your drawing.

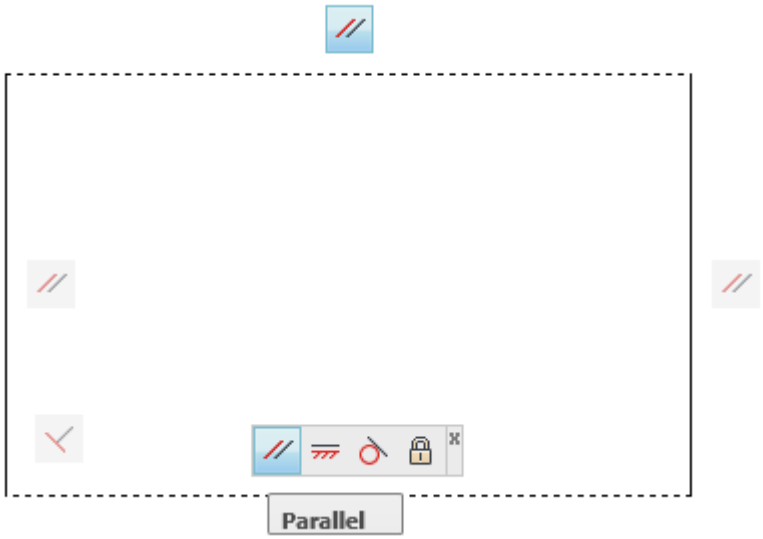


Constraint Bars look and behave a lot like transparent floating tool bars directly in your drawing, except that each button on a bar represents a single geometric constraint. AutoCAD places the bars next to the geometry containing the constraint.

When you roll-over a constraint bar with the mouse a small “grip bar” appears on the right which you can use to reposition the constraint bar or you can close the bar by clicking the “X”.

When you roll-over individual constraints on a constraint bar AutoCAD highlights the button, the entity the constraint applies to, and the corresponding button and entity participating in the constraint (if any).

When you right-click on a constraint AutoCAD presents you with a series of operations which you can perform on the constraint, including deleting the constraint, hiding the bar, or managing the constraint bar settings.



In the example at right the user has rolled over the parallel constraint button on the bottom constraint bar. AutoCAD has highlighted the horizontal line at the bottom which the constraint applies to, as well as the other line and constraint bar that participates in the parallel geometric constraint.

You remove all constraints on an entity using the DELCONSTRAINT command. You cannot remove individual constraints using the DELCONSTRAINT command.

Dimensional Constraints

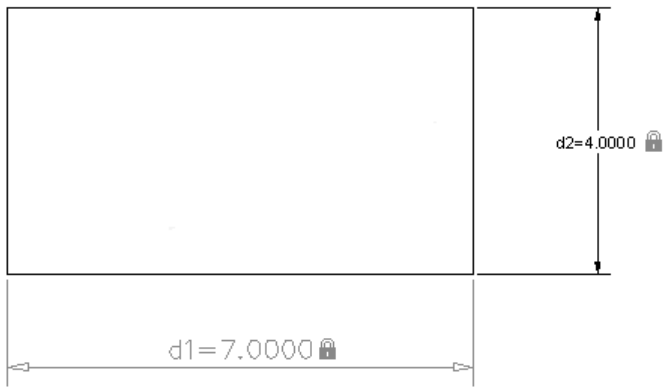
Dimensional constraints define constraints between geometry in your drawing that look and behave a lot like dimension entities:

- Aligned
- Horizontal
- Vertical
- Radial
- Diameter
- Angular

Some dimensional constraints can either be applied to a single entity (e.g. length of a line, radius of a circle) or between entities (e.g. distance or angle between two lines).

Applying dimensional constraints

You use the DIMCONSTRAINT command to create new dimensional constraints. The command prompts you for the type of constraint and the object(s) to constrain.



You also use the DIMCONSTRAINT command to convert existing associative dimension entities into dimensional constraints by selecting the associative dimension instead of specifying a constraint type.

When applying dimensional constraints AutoCAD modifies the constrained geometry to satisfy the new constraint. If more than one entity participates in the constraint, AutoCAD modifies the second entity (if possible) to satisfy the constraint.

When you define a dimensional constraint you also define a *constraint variable name* and an expression. AutoCAD uses the expression to compute the value of the variable. The expression may be a constant or a mathematical expression calculated from other constraint variables. The value of the constraint variable defines the length of the dimension or angle and thereby controls the geometric properties of the geometry.

Reference Constraints

Normally dimensional constraints *define* dimensions of or between entities in your drawing. You create dimensional constraints that *measure* geometric properties of entities in your drawing by changing the “Reference” property of the constraint from “No” to “Yes” in the Properties dialog. Constraints that measure the geometric properties of geometry are called *reference constraints* and are conceptually just like associative dimensions. AutoCAD encloses the text of reference constraints in parenthesis to distinguish them from constraints that control geometry. AutoCAD automatically creates reference constraints if adding the dimensional constraint would over-constrain the geometry.

Use reference constraints to measure distances and report them in constraint variables.

Managing dimensional constraints

By default AutoCAD displays the variable name and expression as the dimension text, along with a lock icon to distinguish regular dimensions from dimensional constraints. You use the CONSTRAINTSETTINGS command to change the behavior to display the variable name and expression, just the variable name, or just the current value, and to show or hide the lock icon.

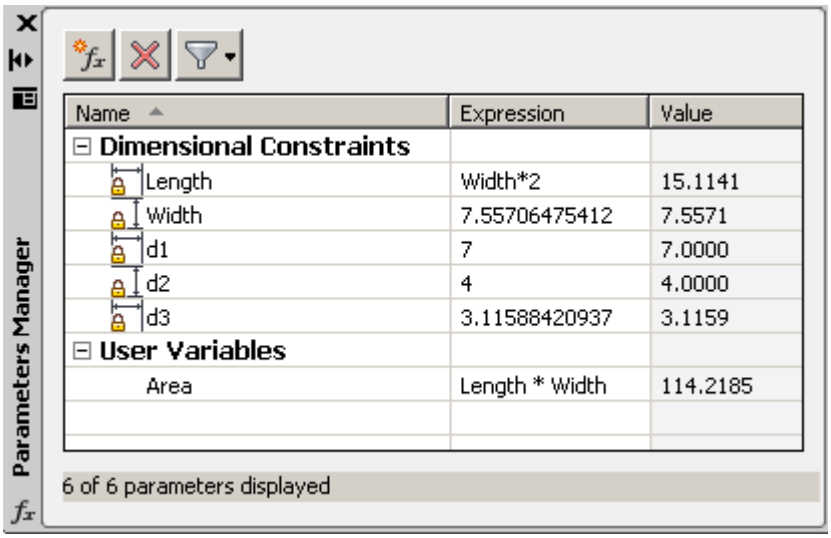
You control when a dimensional constraint appears in your drawing by defining the *constraint form*.

Annotational dimensional constraints look and behave just like regular dimension entities. You control the appearance of annotational dimensional constraints using dimension styles, and they can even dynamically scale if you define them to be annotatively scaled or by using an annotative dimension style. You place them on specific layers and turn the layer on or off to control their visibility, just like normal dimension entities.

Dynamic dimensional constraints do not behave like regular dimension entities. Dynamic dimensional constraints always appear at the same size on the screen (zoom invariant), and always appear in a "dimmed" color. You cannot place dynamic constraints on user-defined layers. AutoCAD normally hides dynamic dimensional constraints until you select an entity with a dynamic dimensional constraint. You manually toggle all dynamic dimensional constraints on or off in your drawing using the DYNCONTSTRAINTDISPLAY command.

You modify the position of dimensional constraints as you would for other dimension entities such as by selecting them and grip-editing their position, or deleting them using the ERASE command. Unlike regular dimension entities AutoCAD only allows you to modify the location of the dimension line and text. To modify the length you modifying the constraint expression via Property Palette, by double-clicking on the text in the editor, by the *Parameters Manager ESW*, or by grip-editing the dimension.

You access the Parameters Manager ESW using the PARAMETERS command. The Parameters Manager ESW provides access to all of the dimensional constraint variables in the current space. From this ESW you can change the names of variables and modify the expression used to calculate the variables. The list includes annotational and dynamic dimensional constraints.



You can also create *user variables* from the Parameters Manager ESW. User variables are constraint variables that do not directly constrain geometry.

User variables provide the ability to compute values from other variables. You can also use user variables in the expressions of dimensional constraints just like a constraint variable.

Establishing Constraints Intelligently: Vertices and Edges

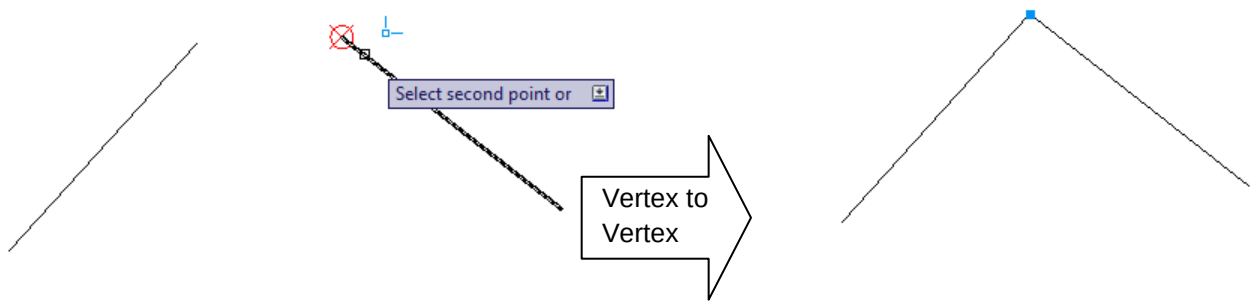
What you select when establishing geometric or dimensional constraints is an important consideration which affects the kind of constraint you establish.

AutoCAD establishes all 2D geometric constraints on a *vertex* or *edge* sub-entities of the entity you select when establishing a constraint. AutoCAD allows some geometric constraints to be established only between two vertices, such as when creating linear dimensional constraints. Other constraints require at least one of the sub-entities to be a vertex, such as the coincident constraint. Some constraint types can only be established between two edges, such as the parallel constraint.

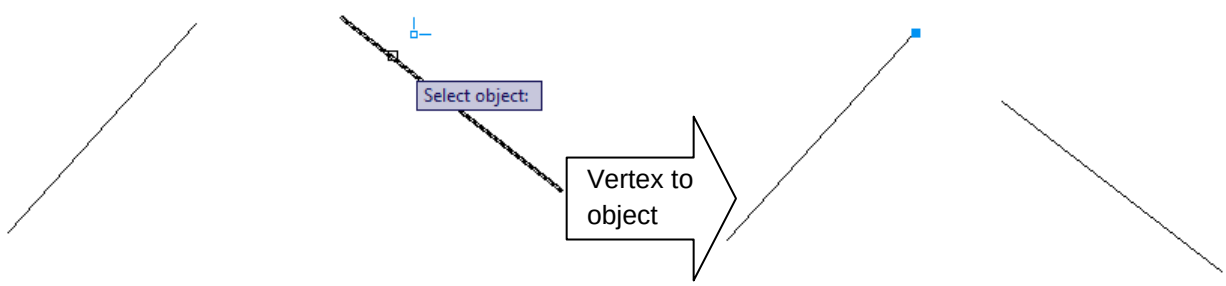
By default AutoCAD selects the nearest vertex on the entity you select by default when the constraint allows both vertex and edge selections. AutoCAD displays a red “bullseye” over the vertex to indicate that you are establishing a constraint to a vertex.

You can change to “edge” selection mode using the “Object”. The command only provides this option if the constraint can be established between a vertex *or* an edge. AutoCAD does not allow you change modes if the constraint only supports edges or vertices.

Suppose you want to establish a coincident constraint between two lines. If you use vertex mode to select the second line you establish a coincident constraint between the end point of the first line and the vertex of the second line:



If you use edge mode to select the second line you establish a coincident constraint between the end point of the first line *and the infinite line defined by the second line*. In this case AutoCAD would allow the first line to “slide” along the axis of the second line when you try to move either line.



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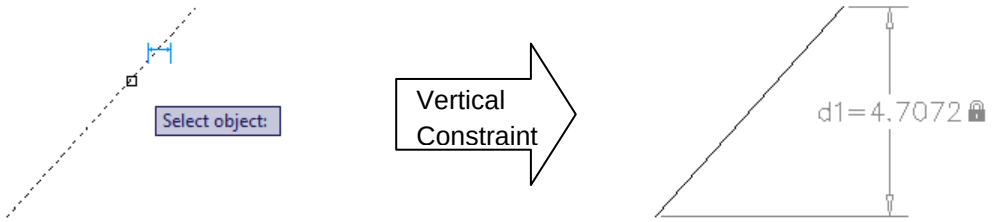
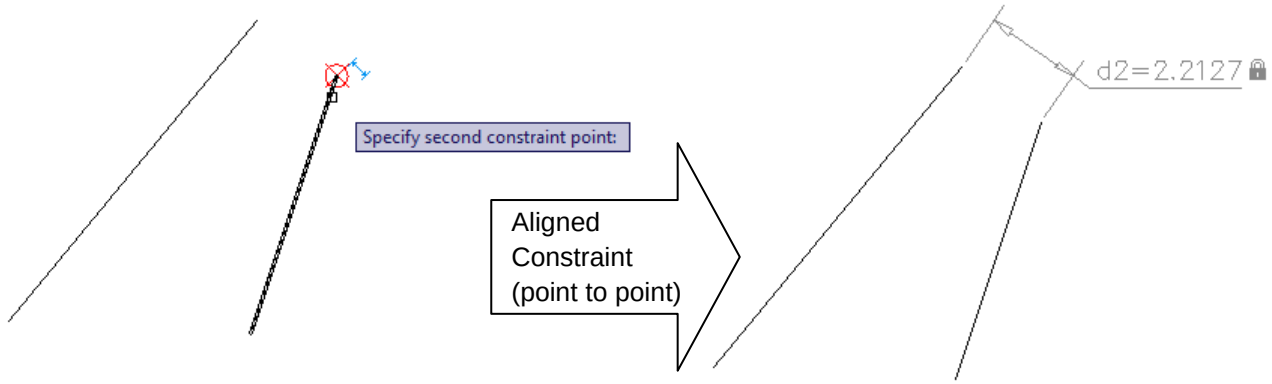
AutoCAD only allows you to select vertices when you create horizontal or vertical dimensional constraints. For these constraint types the “Object” option allows you to quickly establish the constraint between the two endpoint vertices of the object you select, it does *not* enable edge selection:

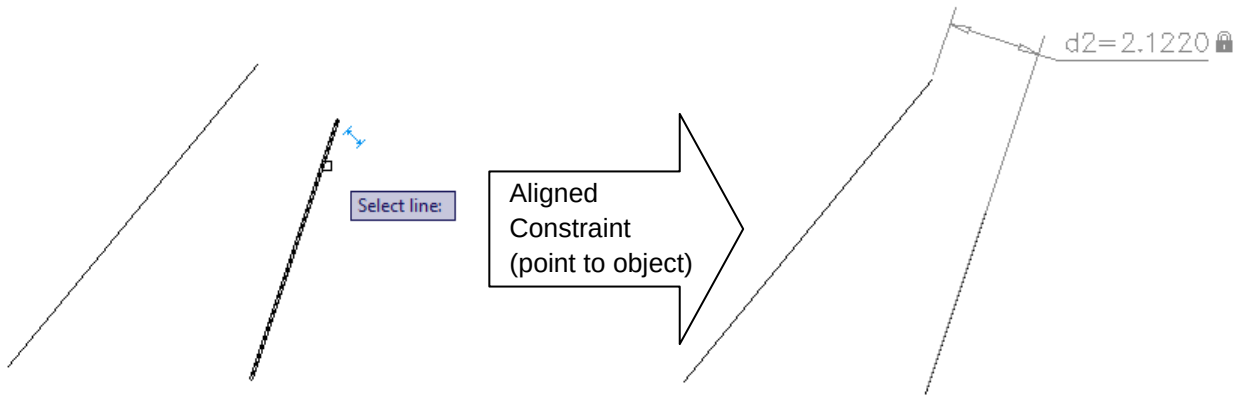


AutoCAD allows several options when creating aligned constraints. You can establish the constraint on a single object (AutoCAD constrains the endpoint vertices), between a point and a line, or between two lines.

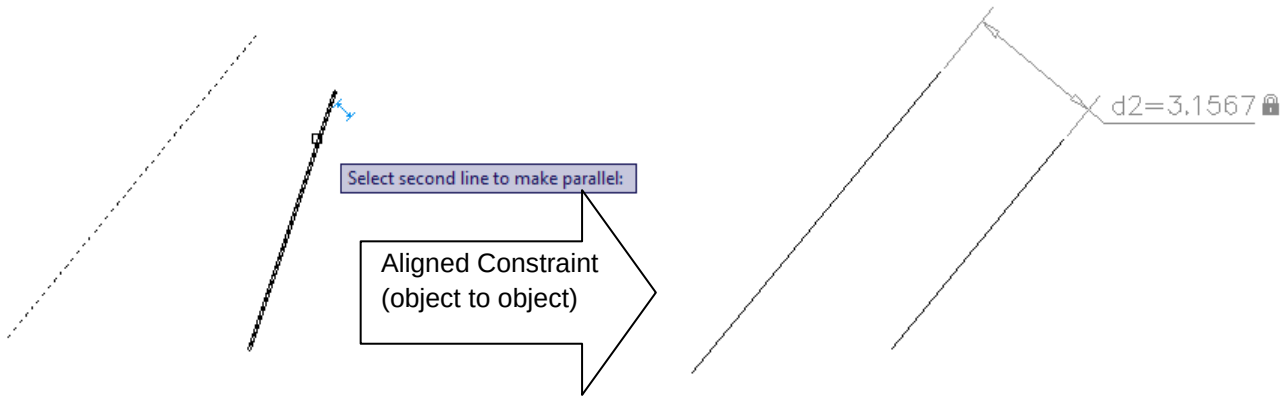
When creating an aligned constraint between the endpoints of two lines AutoCAD places the constraint perpendicular to the first line:



When creating an aligned constraint between a point and an edge AutoCAD positions the constraint perpendicular to the selected edge:



When creating an aligned constraint between two lines AutoCAD makes the second line parallel to the first and places the constraint perpendicular to both edges:



Establishing Constraints Intelligently: Properly Constraining Geometry

One of the biggest causes of confusion and frustration when working with constrained geometry arises from under constraining the geometry. When you modify under constrained geometry AutoCAD produces just one of possibly an infinite number of configurations which satisfy the constraints. When AutoCAD produces a different configuration than you expected is usually a hint that you need to apply more constraints. Understanding

Each 2D entity in your drawing has a certain number of *degrees of freedom* which define its shape and orientation. For example with a circle you can change its center point X or Y, or its' radius, so it has three

degrees of freedom. For a line you can change the X or Y coordinates of the start and end points, so it has four degrees of freedom.[1]

When you apply constraints to an entity you remove one or more degrees of freedom. For example a fixed constraint defines the X, Y coordinate of a point thereby removing two degrees of freedom. The number of degrees of freedom remaining on an entity determines its constraint status.

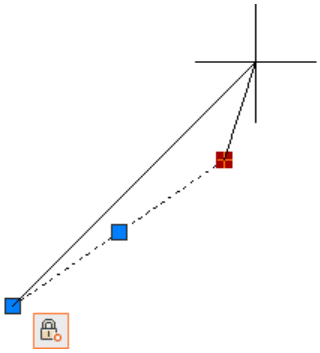
- *Under constrained* refers to an entity whose number of degrees of freedom is nonzero.
- *Well constrained* (or *fully constrained)* refers to an entity whose number of remaining degrees of freedom is exactly zero.
- *Over constrained* refers to an entity with more constraints applied than degrees of freedom. AutoCAD refuses to apply any constraints that result in over constrained geometry.

Under constrained geometry may respond to normal editing operations, such as grip-editing or other commands that modify geometry, depending on the constraints applied to it and the nature of the operation you perform.

Fully constrained geometry does not respond to grip editing or to commands like MOVE, ROTATE, or STRETCH, because the geometry is completely described by the constraints. Many people confuse well constrained geometry with over constrained geometry. To modify well constrained geometry you must modify the constraints themselves, for example by changing the value of a dimensional constraint or by relaxing the geometric constraints while grip editing.

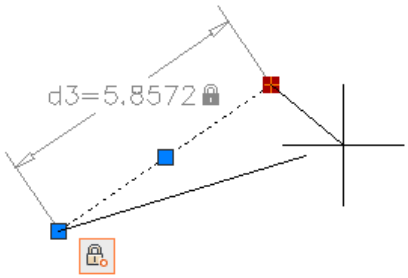
Tip: You relax constraints while grip editing by pressing and releasing the CTRL key. Pressing it again turns constraints back on. AutoCAD doesn't enforce any constraints when grip editing with constraints relaxed. When you complete the editing operations AutoCAD automatically deletes any geometric constraints that are no longer satisfied and updates dimensional constraints based on the modified geometry.

Consider a line entity, which normally has four degrees of freedom. When you apply a fixed constraint to one end of the line you remove two degrees of freedom, but two remain so it is under constrained and you can still directly modify the other endpoint of the line.

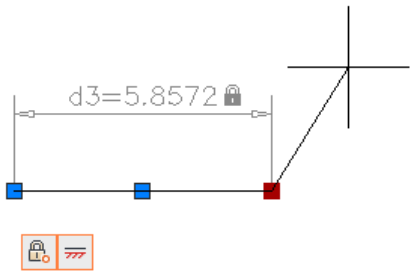


[1] When limited to a 2D plane these are the degrees of freedom on a line and circle. In 3D there are three more degrees of freedom: the Z co-ordinate, and rotation about the X and Y axes, together increasing to six the number of degrees of freedom on a circle or line. AutoCAD's constraint system only supports 2D planar entities effectively fixing these degrees of freedom and removing them from consideration.

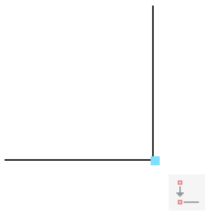
When you add an aligned constraint to the line you remove one more degree of freedom (the length of the line). You can still modify the line by rotating it about the fixed endpoint.



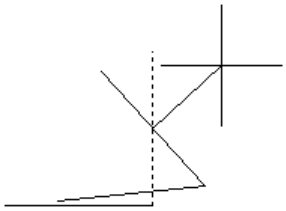
To fully constrain the line you need to remove one more degree of freedom by fixing its orientation relative to the UCS. You might apply an angular dimensional constraint or a horizontal geometric constraint. To modify the fully constrained line you must change the value of the dimensional constraint “d3”, or used relaxed constraints while grip editing.



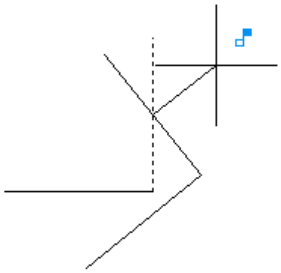
Things get more complicated when you try to constrain groups of entities, since there are more degrees of freedom to remove. Consider two lines which are initially perpendicular and share endpoints having a coincident constraint. When you rotate one line about its unconstrained endpoint AutoCAD updates the other line to maintain the coincident constraint. There are many possible and equally valid solutions. To maintain the coincidence of the endpoints AutoCAD might *move, rotate,* or *stretch*, the other line, or any combination of modifications which satisfy the constraints. The following figure shows the initial geometry and just two of the possible solutions:



Starting position with a coincident constraint at the shared end point.

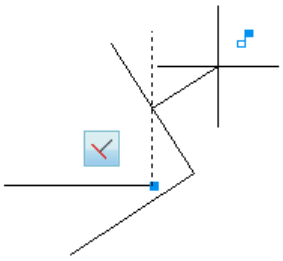


One solution: AutoCAD moves and rotate the horizontal line when you rotate the vertical line.



Second solution: AutoCAD rotates the horizontal line when you rotate the vertical line.

In fact AutoCAD provides the first solution with this geometry, but many people may find the second solution “more desirable”. When you expect one result and AutoCAD provided a different one you usually need to add constraints.



Adding a perpendicular constraint between the lines removes an additional degree of freedom and enforces the desired relationship. Note that the geometry is still under constrained, for example both lines could change length.

Using Constraints in Dynamic Blocks

You use dimensional and geometric constraints in dynamic blocks by adding them to the block definition when you create the rest of the geometry in the block editor.

AutoCAD does not allow you to share constraint or user variables between model (or paper) space and a block definition. Model space variables cannot be referenced from within a block definition, and variables in a block definition cannot be referenced from model space.

AutoCAD allows you to expose constraint variables as custom properties of a block by defining the variable as a *constraint parameter* or as a *user parameter*.

Using constraint parameters

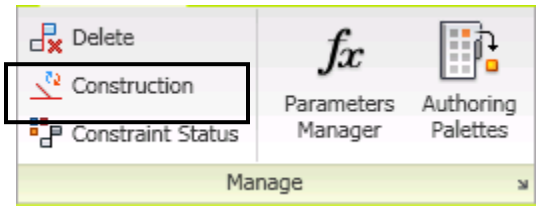
Constraint parameters combine the custom properties and grips of an *action parameter* with a *dimensional constraint*. You add constraint parameters to dynamic blocks to define dimensional constraints with custom grips and properties which allow manipulation of the geometry in the block.

You define constraint parameters using the BCPARAMETER command which combines features of the DIMCONSTRAINT and BPARAMETER commands. You create constraints between vertices and edges of geometry in the block. Adding a parameter defines a new parameter variable and an expression. You can also convert existing dimensional constraints directly into constraint parameters.

The only significant differences between constraint parameters and dimensional constraints are:

- Constraint parameters define variables which act as custom properties of the block. You modify the properties by selecting the block and using Property Palette to change the property value.
- Constraint parameters define custom grips that you use to grip-edit the block.

Using construction geometry



Construction geoemtry helps to maintain geometric and dimensional constraints in your dynamic blocks. You create construction geometry using the BCONSTRCTION command.

Traditionally you managed construction geometry using special layers where you placed the construction geometry and controlled the visibility of the geometry by managing the layers. While functional this approach takes time to maintain and can lead to errors in your printed documents when layers are left on, etc.

With the BCONSTRUCTION command you convert regular entities into construction entities, and you can revert constcution entities back into regular entities. You also use the command to show or hide all construction geometry in the block.

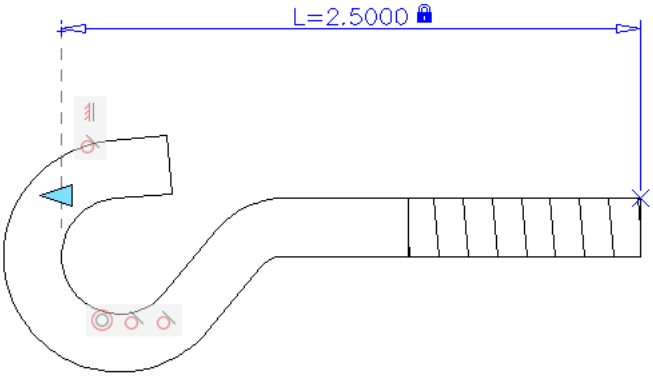
AutoCAD manages the construction geometry for you:

- AutoCAD draws the geometry with a hidden line type and with a “dimmed” color
- AutoCAD keeps construction geometry on layer 0. If you change the layer of a construction entity AutoCAD puts it back on layer 0 the next time you open the block in the editor.
- AutoCAD makes construction geometry visible when you open a block in the block editor.

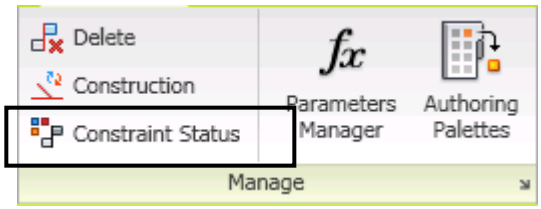
Tip: Think of construction geometry as an invisible “skeleton” that you use to control the behavior of the visible geometry in the block.

Use construction geometry for hidden symmetry lines, to measure distances to fixed points in the block, or to act as “levers” on other geometry that you could not easily constrain otherwise.

The example at right uses construction geometry to measure the distance from the end of a bolt to the opposite side of a bend. A tangent constraint between the construction line and the arc causes it to act as a lever between the dimensional constraint and the arc. A vertical constraint on the construction line keeps it properly aligned, even if the radius of the arc changes.



Identifying constrained geometry

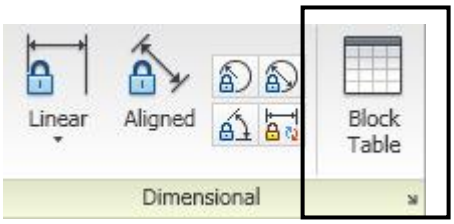


When you turn on the BCONSTATUSMODE system variable AutoCAD display each entity in your block in one of four colors:

White/Black:	No constraints applied
Blue:	Under constrained
Magenta:	Fully constrained
Red:	Overly constrained

This gives you a visual indication of the constraint status of each entity in your block. AutoCAD automatically updates the colors as you add or remove constraints. Use the BSETTTINGS dialog to change the colors used by BCONSTATUSMODE.

Using the block properties table



Expressions allow you to define relationships between parameters in your dynamic blocks. Expressions don't allow you to define *allowed combinations* of properties values in a dynamic block.

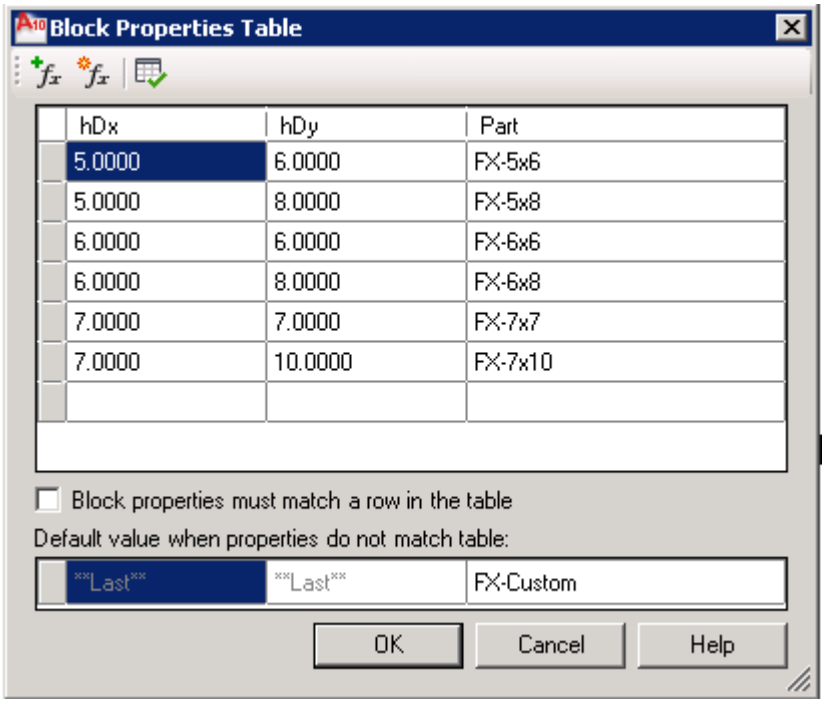
Adding a *Block Properties Table* to your block defines allowed combinations of property values for some or all of the properties in the block. The Block Properties Table is an enhanced version of the Lookup Table available in earlier releases of AutoCAD.

Blocks contain only a single block properties table.

With the Block Properties Table you can:

- Define combinations of values for block action parameters, constraint variables, and attributes.
- Restrict the allowed property combinations to those that appear in the table or allow combinations that do not appear in the table.
- Set all of the properties at once on an instance of the block by displaying the table and picking the desired row from the table or by clicking on the grip to display the combinations in a cascading pop-up menu.

To create a Block Properties Table click on the “Block Table” button in the ribbon or invoke the BTABLE command. AutoCAD displays the Block Properties Table dialog to which you can add rows and columns. Columns of the table represent properties of the block, such as block action parameter variables, parameter constraint variables, user variables, or attributes. Rows of the table define allowed combinations of these property values, with the cells containing the property values themselves.



Click on the “Add Properties” button to add existing parameters, variables, or attributes as new columns to the table.

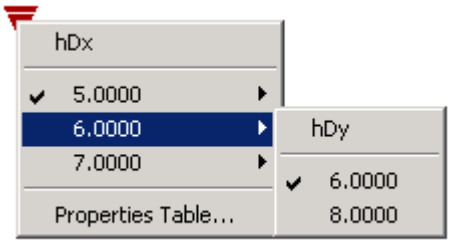
Click on the “New Properties” button to define a new user variable and add it as a column to the table.

Click on a cell to type in a value. If the column represents a constraint variable with an expression, AutoCAD automatically computes values for cells in columns of the table that represent constraint variables defined with an expression. These cells are read-only and cannot be modified.

Click on the label for a column and drag to re-order columns in the table. Click the checkbox at the bottom of the table to restrict all property values to combinations that appear as a row in the table. When unchecked, AutoCAD allows you to specify any combinations of properties for the block. You define the value to assign to each property when a row cannot be matched in the cells at the bottom of the dialog.

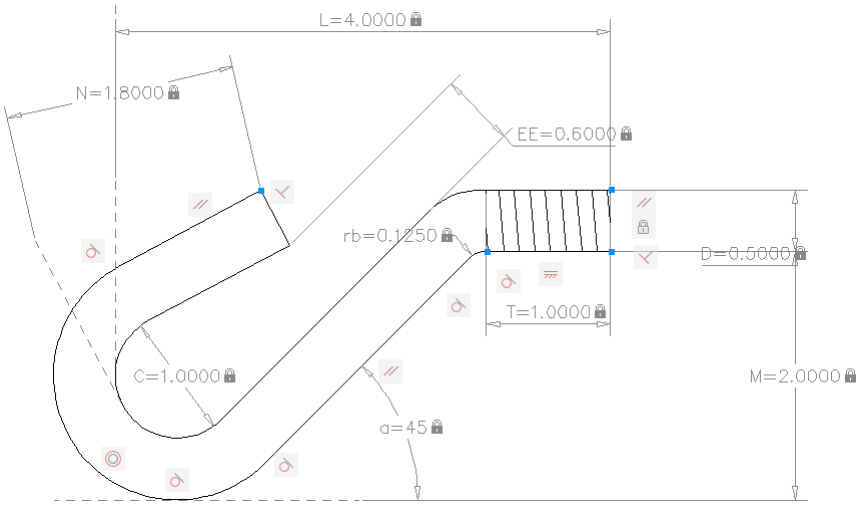
To display the table for a block that you have inserted, click the “Block Properties Table” property in the Properties dialog.

You can also select the block and click the table’s custom grip to select a row from the table. AutoCAD displays a pop-up list containing the values from the first column in the table. If any values in the first column are repeated AutoCAD displays a sub-list containing properties from the second column, and so on, allowing you to choose the desired property combination without displaying the entire table.



Using Constraints with AutoCAD Mechanical

You can use parametric drawings and blocks you create in AutoCAD directly in AutoCAD Mechanical. All of the commands to create and manage constraints, including constraint bars and the Parameters Manager, are available in AutoCAD Mechanical.



AutoCAD Mechanical takes dynamic blocks to the next level with its Content Library feature. Content Libraries provide an organization to your part library that you use to add content to your Mechanical drawings. You create AutoCAD Mechanical content using the Content Editor which is very similar to the AutoCAD Block Editor.

You can create AutoCAD Mechanical content directly from a dynamic block created in either AutoCAD or AutoCAD Mechanical. When converting a dynamic block to AutoCAD Mechanical content, AutoCAD Mechanical:

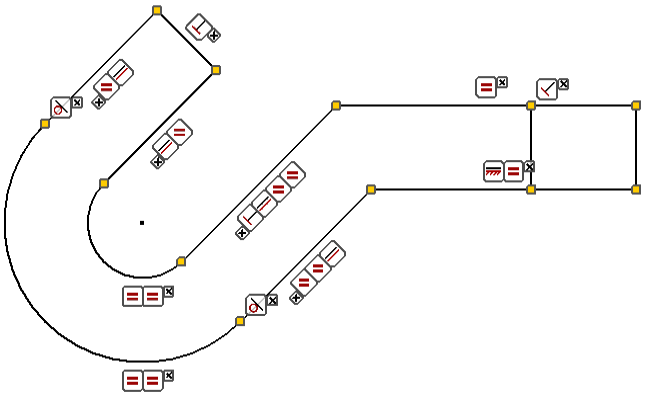
- Retains geometric constraints
- Retains dimensional constraints (both dynamic and annotative)
- Converts parametric constraints to dynamic dimensional constraints
- Converts the Block Property Table into an AutoCAD Mechanical Family Table (similar to but more powerful than the Block Property Table)

AutoCAD Mechanical also offers several powerful tools that can only be used in Mechanical Content:

- Content hatch (AMCHATCH) creates associative hatch whose properties can be defined using an expression (i.e. scale or rotation as a function of other constraint variables)
- Content array (ACMARRAY) arrays geometry in the component using settings that can be defined using expressions (i.e. number of rows and columns, or row and column spacing as a function of other constraint varaibles).

Constriants in Autodesk Inventor

Although Autodesk Inventor can view AutoCAD drawing files, and can be linked to AutoCAD Mechanical drawings for “live updates” via the Inventor Link feature in AutoCAD Mechanical, you cannot directly use constraints or dynamic blocks created with AutoCAD or AutoCAD Mechanical in Autodesk Inventor.



Autodesk Inventor has a separate constraint system built into its 2D Sketching environment, including dimensional and geometric constraints.

Inventor’s 2D Sketching constraints share much of the look and feel of AutoCAD’s 2D constraint system, so transitioning to Inventor constraints from AutoCAD constraints should natural. For example Inventor displays geometric constraints on geometry using icons similar to AutoCAD’s constraint bars, and uses color to indicate the constraint status of the geometry in the sketch.

Inventor’s 2D sketching system offers some advantages over AutoCAD’s. For example Inventor’s “Automatic Dimension” feature is more powerful than AutoCAD’s AUTOCONSTRAIN command, applying *both* geometric and dimensional constraints to the geometry in the sketch. Inventor also automatically applies geometric constraints as you draw in a 2D sketch, saving the need to apply the constraints as a second step.

Constraints in Earlier releases of AutoCAD

Earlier releases of AutoCAD do not maintain the constraints you create with AutoCAD 2010 or later. Earlier releases of AutoCAD effectively operate in relaxed constraint mode when you open a drawing created using AutoCAD 2010:

- You can freely modify constrained geometry.
- Annotative constraints behave as regular dimension entities.
- Dynamic constraints do not appear in the drawing.

When you open a drawing in AutoCAD 2010 that was modified by an earlier release, AutoCAD removes any constraints that have become invalid.

Earlier releases of AutoCAD treat dynamic blocks containing constraints or a block properties table as anonymous blocks.

- AutoCAD hides the block’s custom properties.
- AutoCAD hides the block’s custom grips.

When you open the drawing again in AutoCAD 2010 or later the blocks behave normally.