



AUTOCAD BLOCK BEST PRACTICES

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PREFACE

Block is one of the most important productivity tools in AutoCAD. By optimizing your blocks, you will find that AutoCAD is not just about drawing lines or just get the drawings done. You can draw lines fast, but productivity is beyond that. It is not just to get the drawings done.

You need to be able to modify drawings easily during the design process. You may be able to finish your drawing very quickly, but you may spend too much time when you are doing revisions. If you do, then you are not productive.

Drawings also should provide necessary information. Furthermore, you will want to add some intelligence so that you can automate some process.

In this e-book, you can learn how to create, automate, and manage your blocks. We are not only discussing features, but we are also talking about productivity concept. Moreover, how you can use blocks to solve problems in AutoCAD.

Prerequisite

This e-book is intended for basic to intermediate AutoCAD users. However, I believe it can give some fresh information to veteran users. You have to at a minimum, know and able to use AutoCAD basic features.

The tutorial files

We include tutorial files with this e-book. Extract the zip file to a location you can access easily. When you find instruction to open a drawing file, you can find it in tutorial files folder.

Download the exercise file here: [AutoCAD Block Best Practices tutorial files](#).

Drawing units in this e-book

This e-book uses the metric unit. However, drawing unit should not be an issue. You do not draw many objects in the exercises. We focus more on productivity concept.

However, if this book instructs you to create a new file, create it using ISO template.

TABLE OF CONTENTS

[Preface](#)

[Prerequisite](#)

[The tutorial files](#)

[Drawing units in this e-book](#)

[Table of Contents](#)

[The Block Basic](#)

[Block advantages](#)

[Creating Block](#)

[Defining block name](#)

[Defining block base point](#)

[Defining objects](#)

[Inserting Block](#)

[Insert block definition in a file](#)

[Inserting a file as a block](#)

[Nested block](#)

[Inserting block from another file](#)

[Working with Units](#)

[Modifying Block](#)

[Disassemble and redefine block](#)

[Using block editor](#)

[Replacing Block](#)

[Attaching Information to Block](#)

[Creating attributes](#)

[Working with Block and Attributes](#)

[Creating block](#)

[Inserting Block](#)

[Changing Attribute Value](#)

[Managing Attributes](#)

[Attributes to Reserve Space for Text](#)

[Exercise: Create a room tag](#)

[Working with Layers in Block](#)

[Layer 0](#)

[Layer properties override](#)

[ByBlock](#)

[Using other layers](#)

[Exercise](#)

[Changing layers](#)

[Changing object properties](#)

[More layers practices](#)

[Annotative Block](#)

[Understanding annotative block](#)

[Creating annotative block](#)

[Using annotative block](#)

[Adding intelligence to your block](#)

[Using visibility states](#)

[Adding objects to current state](#)

[Controlling visibility states](#)

[Changing parameter name](#)

[Using parameters and actions](#)

[Adding parameters](#)

[Link action to parameter](#)

[Testing block](#)

[Parameter properties](#)

[Linking stretch action to opposite direction](#)

[Limiting size using list](#)

[Create a list of column size](#)

[Controlling block with constraints](#)

[Using dimensional constraint](#)

[Changing parameters value](#)

[Using geometric constraint](#)

[Using Auto Constrain](#)

[Using block table](#)

[More about dynamic blocks](#)

[Extracting Data](#)

[Information in your blocks](#)

[Attributes](#)

[Parameters](#)

[Using fields in Attributes](#)

[Using data extraction](#)

[First page: Begin](#)

[Second page: Define data source](#)

[Third page: Select objects](#)

[Fourth page: Select properties](#)

[Fifth page: Refine data](#)

[Page 6: Choose output](#)

[Page 7: Table style](#)

[Page 8: Finish](#)

[Updating data extraction](#)

[Linking data extraction to external excel file](#)

[More Block Best Practices Samples](#)

[Example 1: Building elements sample](#)

[Example 2: Automatic Aligning Valve](#)

[Alignment parameter](#)

[Wipeout](#)

[Example 3: Foundation pile cap](#)

[Add reference points](#)

[Adding attributes](#)

[Example 4: Coordinate label](#)

[Example 5: Adjustable table length](#)

[Managing and Sharing Blocks](#)

[Managing block files](#)

[Single block in single file](#)

[Multiple blocks in single file](#)

[Creating and using tool palettes](#)

[Adding blocks to tool palette](#)

[Creating new palette](#)

[Creating palette using Design Center](#)

[Creating palette group](#)

[Sharing your blocks to your team](#)

[After your finish this book](#)

THE BLOCK BASIC

We all learned how to use AutoCAD from different resources. Not all of those resources include block in their learning material. At least I often do not find it in books from local publishers here. I also remember when I took an AutoCAD class; they do not teach me anything about the AutoCAD block. Probably because I only took a basic class.

I assume that this could happen to you too. So let's start this book from the basic of AutoCAD block.

If you already know how to create, insert, and modify the block, you may skip this chapter. However, I recommend you to at least skim through this chapter.

Block advantages

Why are we using blocks?

1. Blocks are single objects. If you have a complex drawing, selecting many lines, hatches, arc, and circles can be tedious and prone to mistakes. If you define them as a block, you can select them easier. You can also select multiple blocks with the same at once using filter or QSELECT.
2. When we update a block definition, it will also update all blocks with the same definition. This feature is very useful if you often need to change your symbol or a common design. However, if you work in a team, you need to consider which part of your design [should use XREF instead of blocks](#).
3. You can save blocks in a separate file as reusable contents library. You easily reuse them later. This technique will make your drawing process faster and also maintain drawing standard. You can share them for all AutoCAD users in your company, to make your drawing uniform between AutoCAD users.
4. Blocks can provide information. You can use that information and report them in a table. For example, the report can be a hole table (in the manufacturing industry), schedule or BOM, or points coordinate.
5. Autodesk has added many features related to blocks. Using dynamic blocks and annotative blocks can help to reduce tasks and confusion among users.

Creating Block

The most basic thing in learning block is to create and to use it. There are several ways to create a block. The common way to create a block is by using create block tool. This tool is located in your ribbon, home tab, block panel.



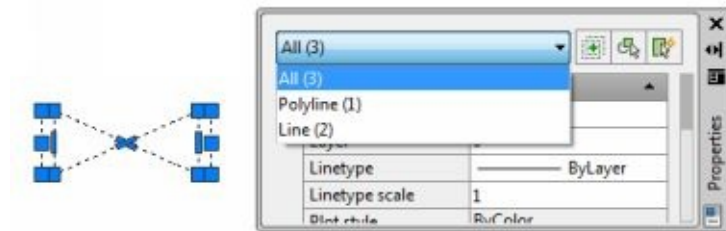
You can also activate “create block” tool by using B then [enter] in command line or dynamic input

Let's try using this tool.

Open valve.dwg included with this e-book.

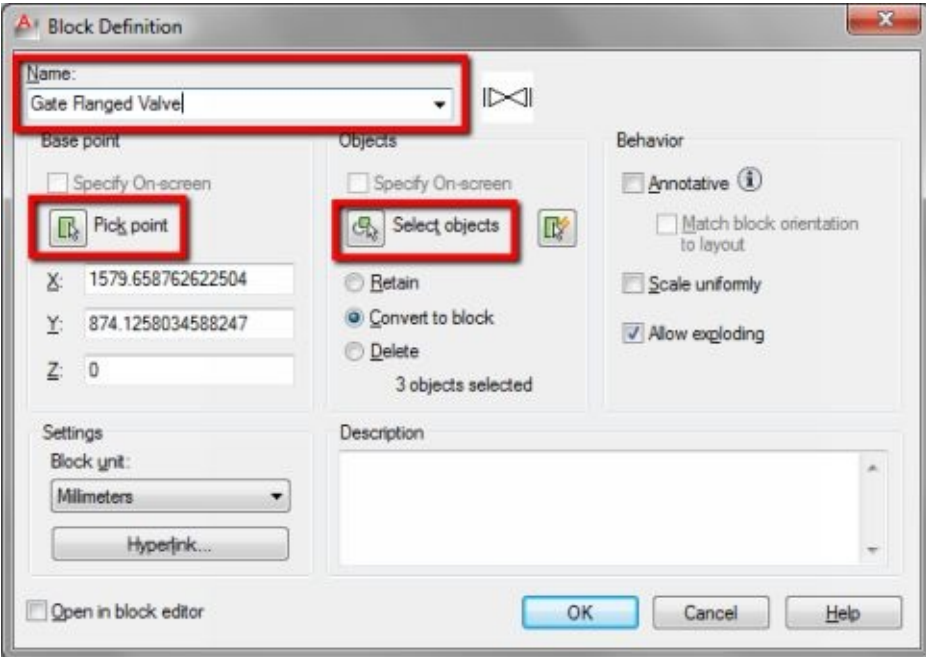


In this file, you can find a valve symbol. This valve is made from one polyline and two lines.



Activate “create block” tool. AutoCAD opens a dialog box.

There are several options available, but let's focus on field and buttons that are shown in the red rectangle.



To create a block, you have to define at least:

- 1. Block name
- 2. Base point of the block
- 3. Objects you want to add to the block.

Defining block name

You have to give a unique name to your block. Do not give the same name with existing block in your drawing, unless you want to replace the definition. We can see [how replacing definition affects your drawing later](#).

Give this block name: *gate flanged valve*.

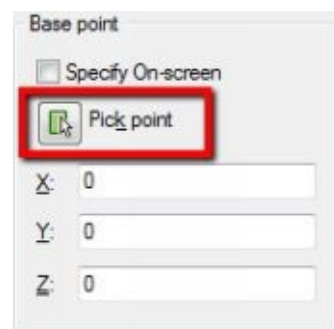


Type it in name field like shown above.

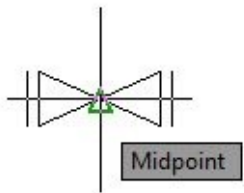
Defining block base point

A block base point is your reference point when you use it. For a valve, we use its center. It makes us easier to snap it to a pipe when we draw a P&ID diagram.

Click pick point button.



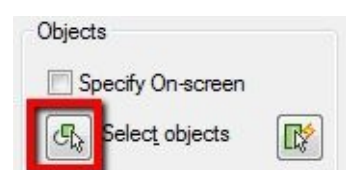
The block dialog is closed temporarily, allowing you to select a point. Snap to midpoint as shown below.



Block definition dialog is restored.

Defining objects

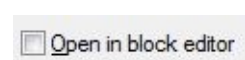
The last thing we need to do is to select objects. Click Select Objects button. This action also closes the dialog box temporarily.



Select all objects then press [enter] to end object selection and bring back block definition dialog box.

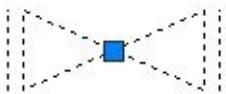
Now you should see the block preview on the right side of block name field.

Before we make the block, make sure open in block editor option is unchecked. We do not use block editor in basic block tutorial.



Now click OK in the dialog box. Save your file.

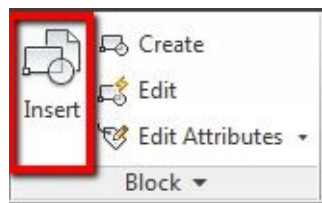
If you select the valve, now it is only one object. No longer consists of three objects. You can only see one grip: the block base point.



Inserting Block

We have created the valve block. Now let's use it.

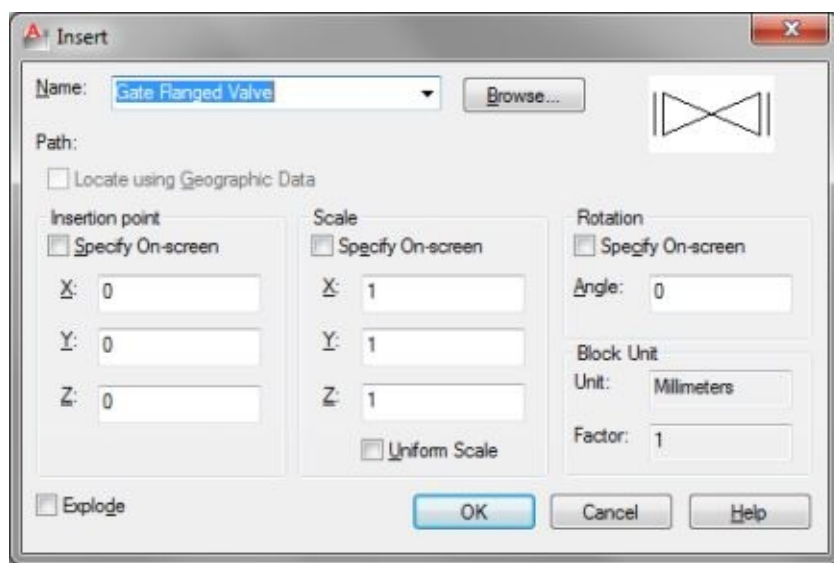
You can insert your block by activating insert block tool. You can find it on *AutoCAD ribbon > home tab > block panel*.



Command alias for insert block is I then [enter]

Insert block definition in a file

After you activate the tool, you can see insert block dialog.



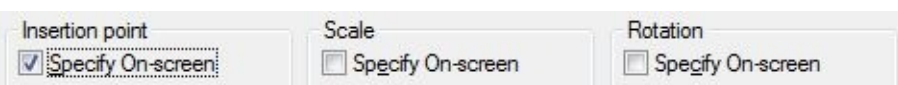
Your block name is in name drop down field. We only have one block here. When you have more block definitions in your file, you can see all the blocks in the list. You can choose which block you want to insert by clicking down arrow on the right side of the name field.

There are three options we can define here: insertion point, scale, and rotation. If you choose to activate “specify insertion point” then you must set the value by clicking points on drawing area.

The common practice is we define insertion point by clicking a point on the screen. However, for scale and rotation would be easier if we set the value in this dialog box.

Let's try to insert it.

Check specify on-screen in insertion point section. Leave the check boxes unchecked for scale and rotation section.



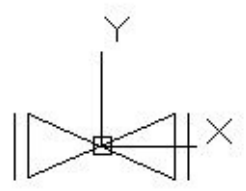
Make sure scale set to 1 and rotation angle to 0. Click OK and click a point to place it. We only need to define one point because we already define scale and rotation in the dialog box.

Try to insert the block again. This time check “specify on screen” for scale and rotation angle. Examine how it works. Don't forget to check your Command Line to see what is AutoCAD asking you to do!

Inserting a file as a block

We can also insert a file into the current drawing. All objects in the drawing file will be converted into a single block. AutoCAD uses the file name as block name.

Open *Valve – Gate Flange.dwg*. The objects are the same with the previous file we used before. It has a polyline and a couple of lines. It is not a block yet.

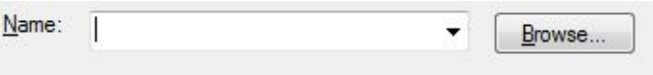


One thing you should remember when inserting a file as a block. The drawing origin (0,0) is the block base point. You can see the UCS showing the block base point. If you want your base point somewhere else, move your objects.

Without making any changes, close this file. Click Cancel if AutoCAD asks you if you want to save it.

Create a new file, then activate insert block tool.

Click Browse button, then select **valve – gate flange.dwg**.



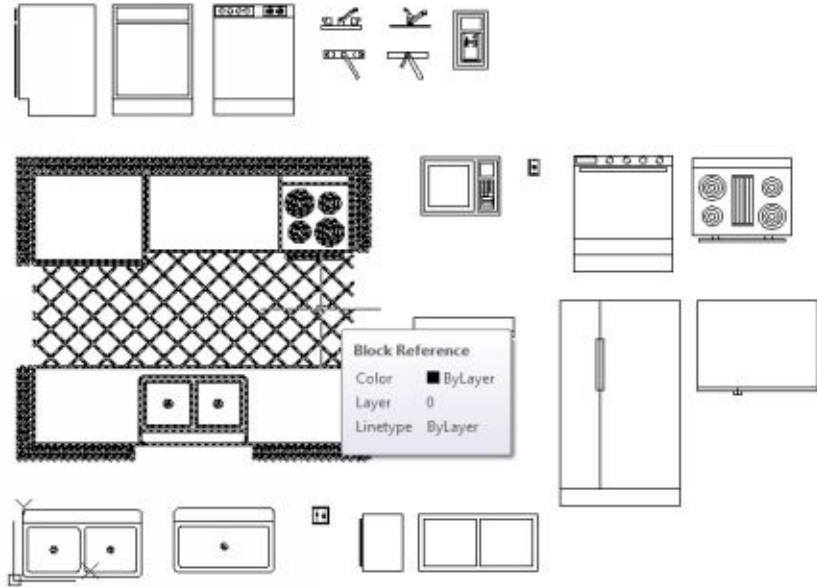
Click open. Now you can see the file name in block name field.

Place your block. Now select it to check if it is defined as a block.

Nested block

A nested block is a block with one or more block definition inside it.

Open *kitchen.dwg*. You can find it in provided files or sample file from AutoCAD installation. This file is a block library. There are several blocks inside it.



Close this file without saving it.

Create a new file, then insert *kitchen.dwg* as a block. Before you insert the file, examine that the block name field is empty.

The whole file becomes one block: *kitchens*. There are several blocks inside our kitchen block. This kitchen block is a nested block.

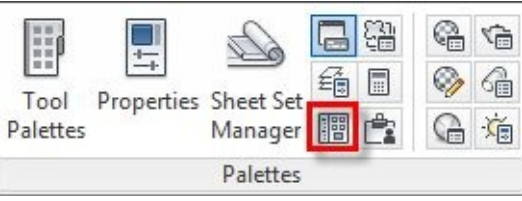
Activate insert block tool one more time. This time you can see not only *kitchens* block is listed here. There are also all blocks in that file.

Inserting block from another file

Block definition is file specific. If you create a block in one file, you only have the block definition in that file. If you create a new drawing file, the block definition is not available there.

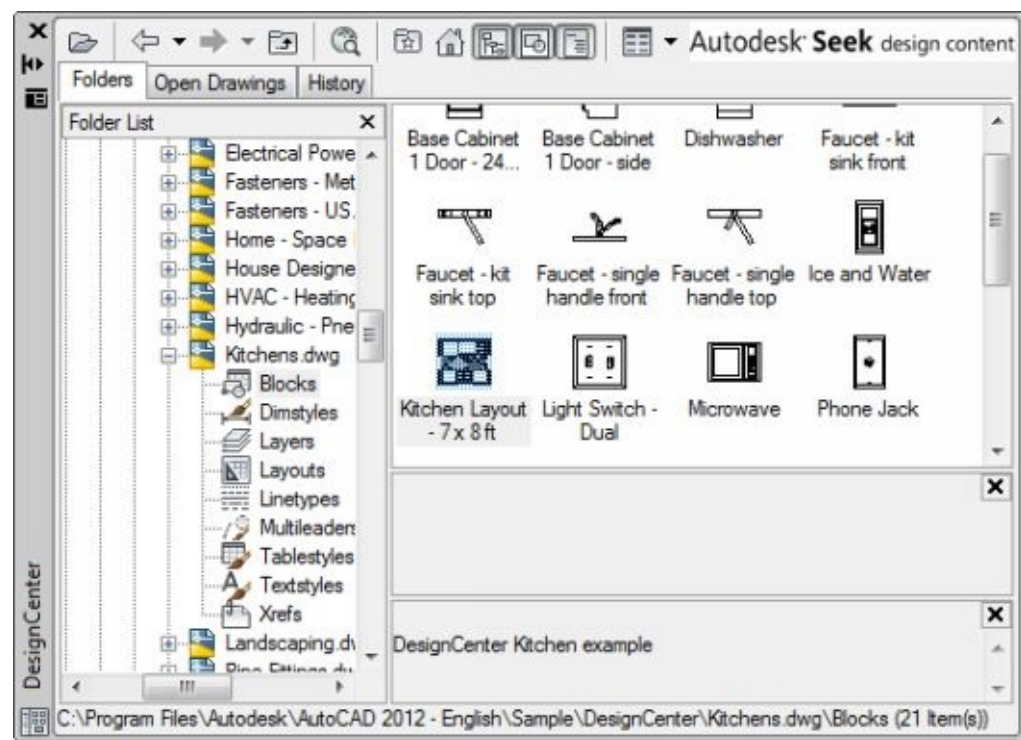
So how can we insert a block from another file? We learned that we could insert a file as a block. If there are several blocks inside the file, AutoCAD also imports all block definitions. You can delete the *kitchens* block and insert only block what you need. However, this is not a good practice. It brings many blocks that you do not need into that file.

A better way to import a block from other file is by using design center. You can open design center palette by accessing it from your *Ribbon > View tab > Palettes panel*.



You can also open it by typing *ADCENTER* from the Command Line or press *[ctrl] + 2*.

The Design Center looks and feels like Windows Explorer. You can see folders and file name in the left pane, and items inside the folder on the right pane. The difference is you can see what is inside a DWG file. You can see blocks, dimension styles, layers, and other drawing styles.

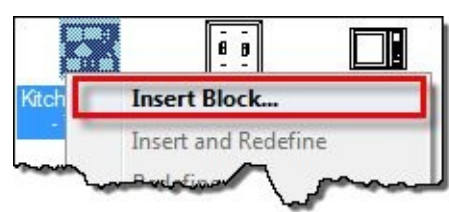


If this is the first time you use Design Center, it should open sample files folder. If you do not see the sample files, navigate to C:\Program Files\AutoCAD xxxx\Sample\DesignCenter.

C is your drive letter where you install AutoCAD. Navigate to another drive if you do not install AutoCAD on C drive. XXXX is the AutoCAD version you have.

Browse and choose any DWG file. Click the + sign next to the file name to expand it. Find *Blocks* category. Click to select it.

The right pane now shows blocks preview. Find a block that you like, right click on that block. Click insert block in the context menu.



AutoCAD inserts your block into the current drawing.

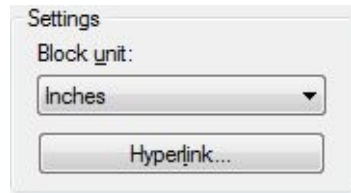
You can also drag and drop the block to your drawing area.

This method inserts only the block you need, not all definition in the drawing.

Working with Units

Before we continue, let's discuss the relationship between drawing and block unit.

When we create a block, there is an option for your block unit.



By default, AutoCAD uses your drawing file unit. Many AutoCAD users do not concern about units, and this can make many problems in the future. If you have not taken this seriously, it is about time that you do.

AutoCAD does not understand real world measurement. When you create a drawing or a block, you need to tell AutoCAD what unit you are using. If you have a block drawn in inches, then you insert it to a drawing with the metric unit (mm), AutoCAD resize it as necessary.

You can convert it manually, but if you set it properly for every blocks and drawing, this can avoid unnecessary mistakes.

To prevent confusion in your drawing, start with an appropriate template. If you draw in metric, use ISO template when you create a new drawing. Imperial users usually do not have this problem because AutoCAD uses the imperial unit as default.

Modifying Block

I consider block as reusable contents. I usually do not modify the block library, but I replace blocks in my drawing.

In real life, when I placed a chair in my room then I want a different model, I do not modify it. I replace it with a different model. The same concept applies to block. If you modify a block, you will have a different object with the same name. Modifying block can be confusing if you place standard parts/objects.

However, many AutoCAD users consider it as a benefit of using block. When you update a block definition, every block instance using the same definition is updated. So this is true if you draw a common design and define it as a block.

I suggest you modify block for these reasons:

1. The block needs correction/update. We usually modify a block in block library for this purpose, not only in the specific drawing.
2. The block defines a common design that we often modify during the design process. However, for standard parts (bolt, nuts, and other standard parts) avoid to modify it. You may need it in the future. Create new block then use the new one.

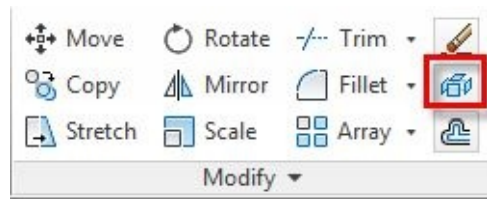
Disassemble and redefine block

This method is the only way we can do it in the past (in old AutoCAD version). We have to explode our block, make necessary changes, then create a block with the same name.

Let's open *valve.dwg* we used before. We have defined a block here. Use Copy tool and make several copies of your block. The number and locations of your blocks do not matter. We just want to see how modifying block affects your drawing.

Select a block, then explode it.

You can find explode on your *Ribbon > Home tab > Modify panel*.

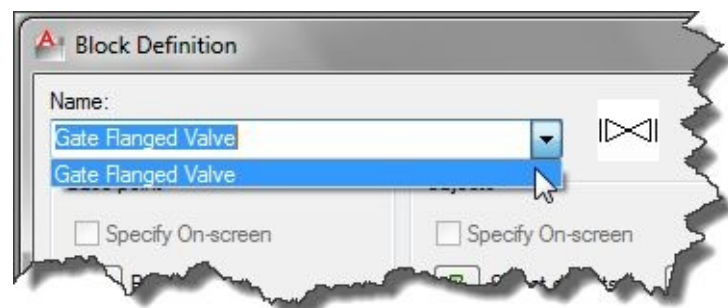


Add two horizontal lines to this valve. Our engineers ask us to modify our block to make it clear where to connect their pipes to our valve.

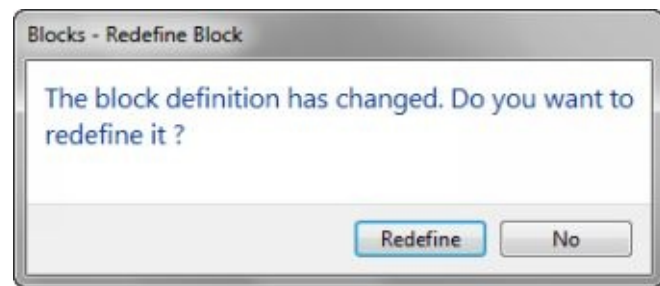
Use image below as a reference.



We finished doing the changes. Activate “*create block*”. This time, we do not have to type block name. Choose existing block name in the dropdown field.



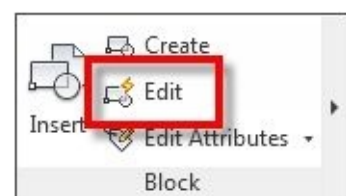
[Repeat the same process for creating AutoCAD block.](#) After you recreate the block, AutoCAD shows a warning. The warning says our action will redefine existing block. It asks us to redefine or cancel the action. We do want to redefine it. Choose redefine.



Examine your drawing. You should see all of your blocks are now updated.

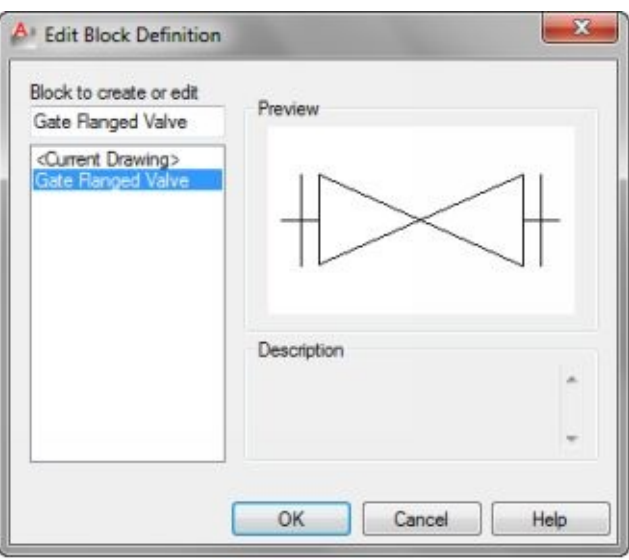
Using block editor

Another way to modify block is by using block editor.



Block editor is a very powerful tool. It is not just we can modify block here, but we can also add intelligence to it. We cover the dynamic block in [the later chapter](#).

Activate edit block tool.



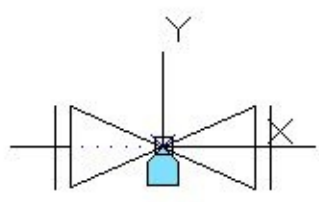
AutoCAD asks you, which block you want to edit. If you have more than one block, you see them listed here. Select *gate flanged valve* and click OK.

Alternatively, you can select a block from your drawing, right click and choose block editor from the contextual menu.

AutoCAD open block editor. You see contextual ribbon tab, only tools you need to modify your block is available here.

You can work with your block as you draw in AutoCAD. Do not move your block from its origin (0,0). Remember, [the origin of your block is its base point](#).

Let's make some changes here. Add more lines to make our valve to look like below.



After you finish, click Save block from your ribbon.



Click close block editor on the right most of our contextual ribbon tab.

AutoCAD asks you to save the changes or discards it. Click save changes.

If you do not save it before you click close, AutoCAD asks you if you want to save your block. Similar to when you exit AutoCAD, but you have not save your file after making some changes.

Replacing Block

We use “*modify block*” to make changes. However, during the design process, we often need to replace objects with other objects from our library.

If we want to replace all blocks with another block definition, then we can use replace block tool. Replace block is an express tool. You need to have express tools installed on your computer.



If you do not have it installed, you can add it later [using this tip from AutoCAD tips blog](#). If you use AutoCAD LT, this tool is not available for you.

To use this tool, you must have replacement block in your drawing. You cannot replace a block definition with a drawing or block defined in another drawing. Insert or create it first before you can use this tool.

After you activate the tool, press = to activate selection tool. Select block that you want to replace. Repeat the process to select the replacement block.



After you finish, you see all blocks with that definition are updated. Replacing block is a very handy tool if you want to replace all screw with different size, replace a different type of windows and for a similar purpose.

ATTACHING INFORMATION TO BLOCK

Now we know how to create a block. We are going further in this chapter to add information to our blocks. We add information to a block using block attributes.

You may consider attribute as a tag. Similar to you tag your belonging, tags give information to anyone who sees it.

Creating attributes

Open door.dwg. This file is a simple door for plan drawings. It is not a block yet. We are going to add information before we define it as a block.

We will add this information:

1. Door model
2. Door manufacturer
3. Door cost

Activate define attributes tool. You need to expand block panel to see it. Click block panel name (with a small arrow pointing down).

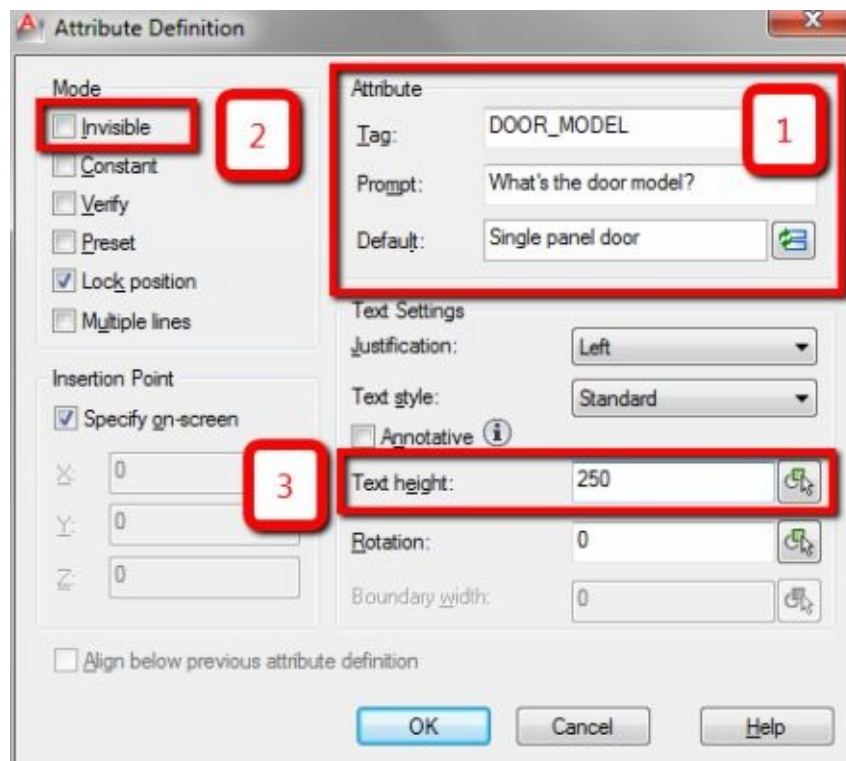
Block ▾

Now you should see the tool. Click the icon to activate it.



You can use ATTDEF in command line to activate it using command alias

AutoCAD opens attribute definition dialog box. For now, let us focus on the highlighted area.



First, let's fill attribute section (1). There are three fields you must give input here.

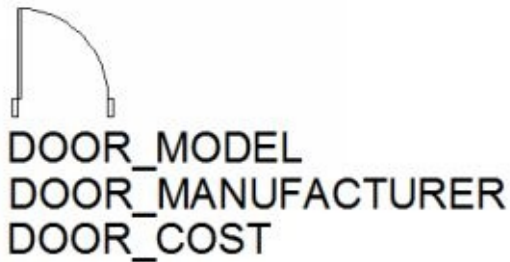
1. Tag is a unique tag name. AutoCAD does not allow you to use space for the tag name. Also, when you use lowercase letters, AutoCAD converts it to uppercase after you create it.
2. Prompt is what users see when they insert the block. AutoCAD asks users what input by showing what you input here. Make sure the question is clear to avoid users providing incorrect data.
3. Default is default value given. If the most answer to this question is 'single panel door', then it makes sense to use it as default value.

We want this information invisible in our drawing. Just for reporting purpose. Check invisible in mode section (2).

Although we do not show the attributes, it is necessary to set text size properly. We may need to modify our block in block editor later. If we set it too small, we might not be able to see it. Change text height to 250. Click OK to finish attribute definition.

Place the attribute under your door.

Create two more attributes: DOOR_MANUFACTURER and DOOR_COST. Make them invisible as well.



Working with Block and Attributes

Creating block

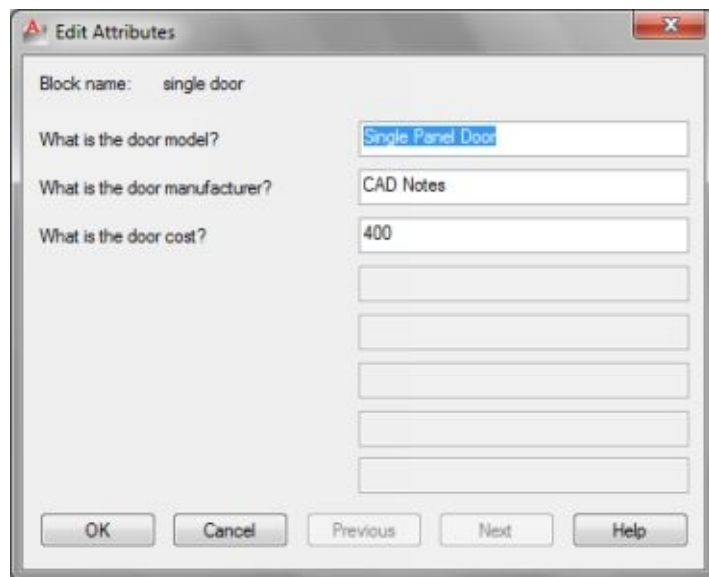
We have finished creating attributes definitions, but we have not created our block yet. Creating a block with attributes is the same process [as we did before](#). The difference is we need to select attributes individually. Selection sequence determines prompt sequence. The first attribute you select is asked first to the users.

Let's try it. Activate “*create block*” tool. Click select object from the dialog box. Select all doors object at once. Then select each attribute individually, from top to bottom.

After you finish selecting them, create the block. Don't forget to give your block name and set its base point.

If you have difficulties, [refer to the previous chapter](#).

The first time we define a block with attributes, AutoCAD opens edit attribute dialog box.



Leave the values as they are. Click OK.

Inserting Block

We have our block definition. Now we can insert it. The process is [the same as we did before](#).

After we place the block, we see AutoCAD asks us to provide data.



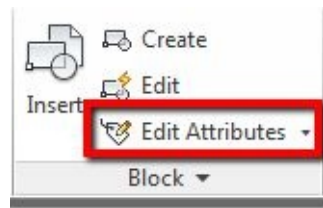
Remember the question and default value provided here? [We made it before!](#)

You can see the questions and fill the value on the command line. However, if you prefer to see a dialog box, change ATTDIA system variable to 1. It allows you to have [the same dialog box like when you first created this block](#).

If you want to keep default value, just press [enter]. However, if you need to change it, type new value then press [enter].

Changing Attribute Value

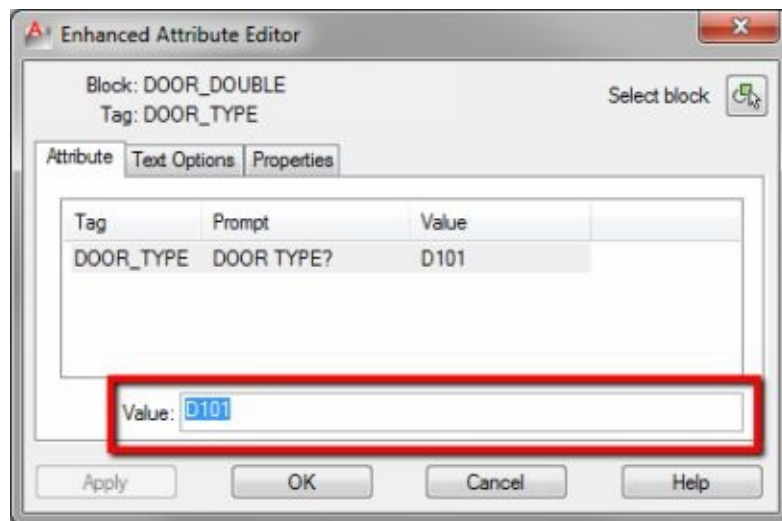
If we need to modify the attribute values, can we do it later? Certainly, you can. We can use edit attributes tool.



You can activate this tool using EATTEDIT from command line OR double-click your block. The veteran users may prefer ATTEDIT or ATE. All of those tools allow you to modify block attributes.

Activate the tool, then select the block you want to edit.

AutoCAD opens a dialog box. Choose the tag in the list and change the value in value field (in red rectangle). After you finish, click OK.



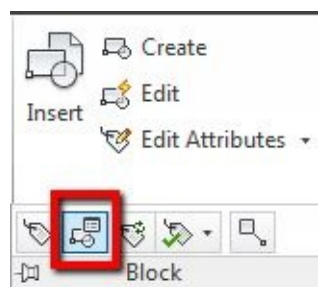
Managing Attributes

We edited attributes value. How can we edit attribute properties?

There are two ways to do this:

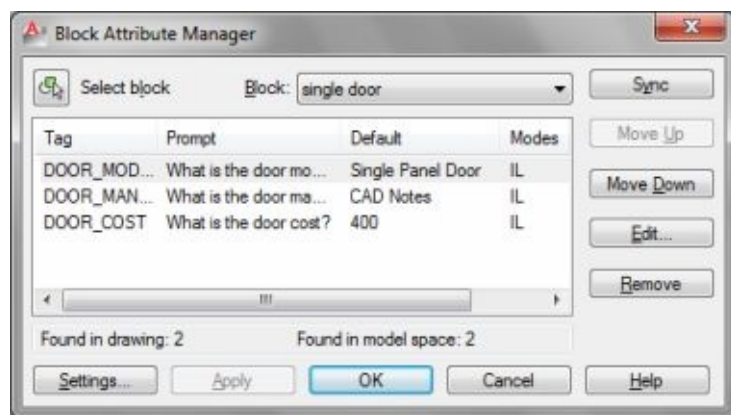
1. Edit block in block editor. Select your attribute then change its properties in the Properties Palette.
2. Use manage attribute tool.

We learn about block editor later. This time we use manage attribute tool. You can activate this tool from *Ribbon > Home tab > Block Panel*.



You can activate “manage attribute” tool by typing BATTMAN then press [enter]

Do you remember when we created a block with attributes? [I mentioned that the attributes selection determines the questions sequence](#). We can arrange attributes sequence, by using this tool. You can press move up/move down to arrange them.



BATTMAN can remove or edit attribute properties too.

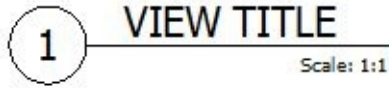
Try the following in manage attribute tool:

1. Select door manufacturer tag, and click move up.
2. Select door model tag and click edit. Uncheck the invisible option. You should see the changes on your screen. Your tag is now shown.
3. Click cancel in all dialog box, to close all dialog boxes.

Attributes to Reserve Space for Text

I believe that attributes were originally meant to add information to block. However, many AutoCAD veterans use block attributes to preserve space for their text. Autodesk also provides some samples that use this method. So it is safe to assume this is a recommended practice.

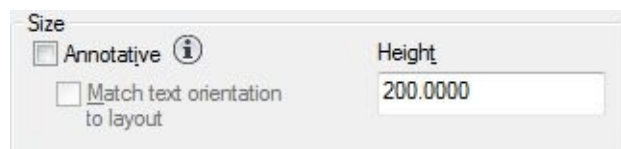
This drawing title is using block attributes. You can see this sample in tool palettes.



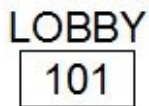
The advantage of using block attributes for this purpose is to maintain the standard. If you defined view title, you could use it for every project. Because you created it once and simply reuse it, you can expect the appearance is the same for every drawing. The font type, font size, and all objects that create the view title are not different. You and your team can use the same block. All you need to do is insert it and change the value of the attribute.

Exercise: Create a room tag

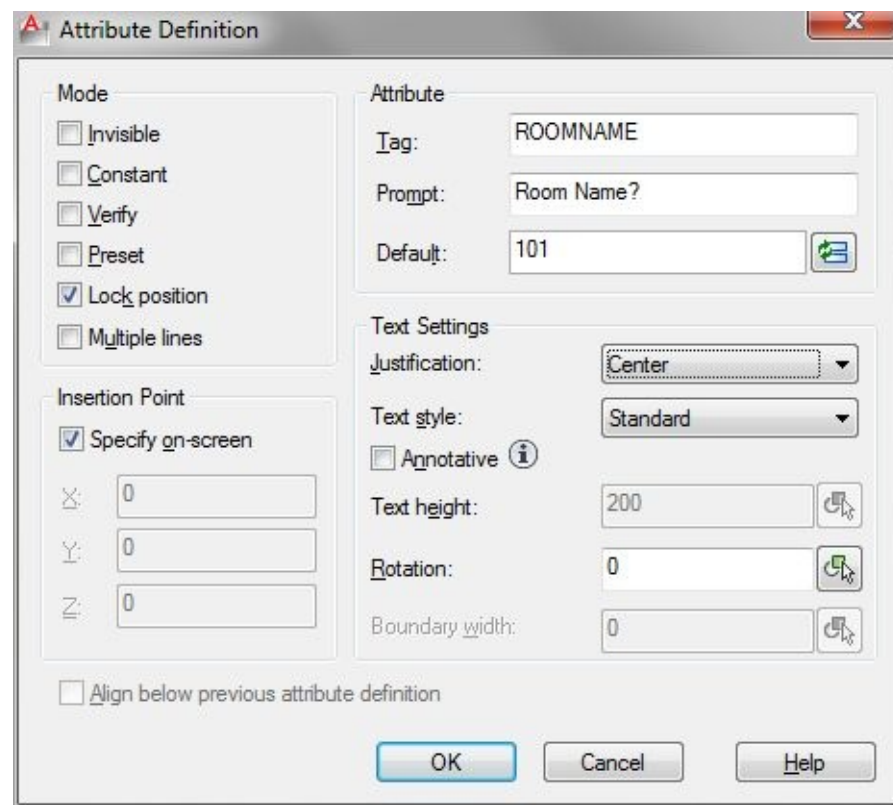
Create a new file. Use acadiso.dwt as your template. Before we start to create our block, open text style dialog. Change standard text height to 200.



We are going to create a room tag. This room tag shows room name and room number.



Activate define attributes (ATTDEF). Set the definition like below.



Make sure the invisible option is disabled. We want this tag to appear in our drawing. Give appropriate tag name, prompt, and default value.

Change text justification to center. Do you see the text height field? Now it is disabled.

If we define text height in text style with 0, then we can change the value when we use it. However, we defined the value to 200 before. Then AutoCAD uses the value every time we use that text style.

Click OK and place your tag.

Create another attribute for the room number. Use the same settings. Don't forget to give it an appropriate name. Place it below the room name.

The last thing we need to do is to add a rectangle around the room number. Create a rectangle like this.

ROOMNAME
ROOMTAG

Create a rectangle that's enough to hold 3 characters inside it. Don't worry if the tag name looks beyond your rectangle. It is supposed to look like that now.

Create it as a block and try to insert it.

You may want to create different annotation block with attributes as exercise.

WORKING WITH LAYERS IN BLOCK

When we want to include objects into a block, we need to consider how we use layers. There are many benefits of using layers. In this chapter, we learn how we can use layers in blocks.

Block can contain objects on different layers. We mostly use Layer 0 as default layer for AutoCAD blocks. We see how other layers work with the block layers.

Layer 0

If you have a simple block, then most likely you draw all objects on layer 0. Layer 0 is a default layer in AutoCAD. Every time you create a drawing file, you find this layer. We cannot delete this layer.

Most people suggest you create objects for a block on layer 0. They advise you to create your objects on layer 0, then make them as a block. When you insert it on a different layer, it inherits the layer properties.

In most cases, it is a good suggestion. However, in some cases, you would want to cheat a little and use another layer. Like in drawings, managing layers can bring many benefits to us.

Layer properties override

Each layer has its properties. The properties are color, line weight, line type, and so on. If you set the layer properties to other than ByLayer, then when you insert the block, it keeps the properties that you set explicitly. This behavior applies if you use layer 0 and any other layers.

ByBlock

Another property you need to understand is ByBlock. Let's imagine block as a container. A block has several objects in it. Each object has its properties. Moreover, they may be on different layers.

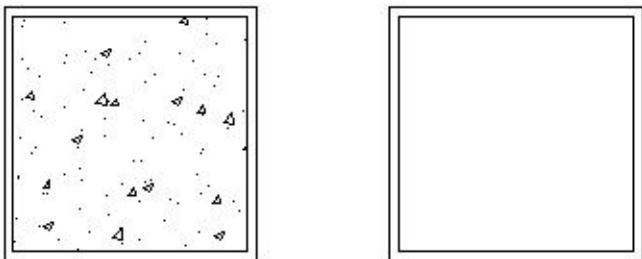
If you set layer properties to ByLayer, it inherits layer properties. When you change the layer color, the color of the objects changes too.

When we set the object properties to ByBlock, then we make it to inherit the block properties. Regardless on which layer the objects are.

Using other layers

Why do we want to use another layer for our objects in blocks? The same reason why we use layers in our drawings.

Let's take this column as an example.



When we plot it in a relatively small scale, we want to show all the details like on the left image. However, when we use the same column in a site plan, the hatch is too dense. We only see a black rectangle. It would make sense to hide the hatches.

Placing the hatch on the different layer allows us to do it.

Exercise

Let's do an exercise to understand this concept better. Create a new file; you may use any template to do this exercise.

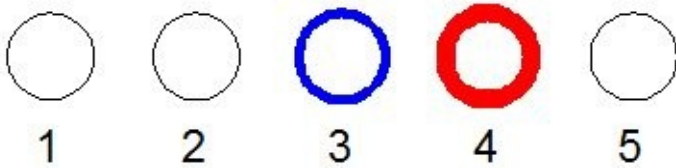
In your new file, add three layers. Set contrast properties for each layer, especially color, line weight and line type.

S..	Name	O..	Fre...	L...	Color	Linetype	Linewei...	Trans.
✓	0	☹	☀	🔒	black	Continu...	— Defa...	0
▢	Layer1	☹	☀	🔒	red	Continu...	█ 1.00 ...	0
▢	Layer2	☹	☀	🔒	green	Continu...	█ 0.30 ...	0
▢	Layer3	☹	☀	🔒	cyan	Continu...	— Defa...	0

Now create five circles. Give the circles number so we can recognize them. I place text numbers on layer 0.

Set these properties for the circles.

Circle	Layer	Color	Lineweight
1	0	ByLayer	ByLayer
2	0	ByBlock	ByBlock
3	0	Blue	0.5
4	Layer1	ByLayer	ByLayer
5	Layer1	ByBlock	ByBlock



You need to make sure show line weight in drafting settings is on. If you do not activate it, you cannot see the line weight difference.

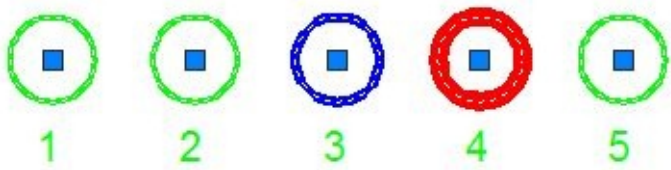


Create a block from each circle. Circle no.1 is the block no. 1; circle no.2 is the block no.2, and so on.

You can open the file that I use in your exercise files > final folder. You may use it without creating your file. However, it is important for you to know how to make that particular block.

Changing layers

Select all circles then change their layer to Layer 2. You can see block 1, 2, and 5 now use layer color and line weight. However, block 3 and 4 are still using their properties.

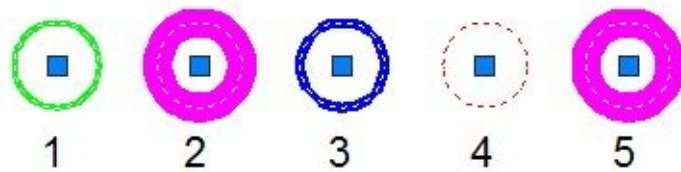


Block Reference (5)	
Layer	Layer2
Name	*VARIES*
Rotation	0

That is easy to understand because we have overridden no. 3 and no.4 properties. They do not follow layer properties.

Changing object properties

Now select all blocks, override their color and line weight.

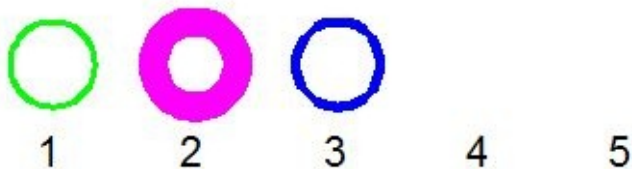


Block Reference (5)	
Color	Magenta
Layer	Layer2
Lineweight	2.00 mm
Name	*VARIES*
Rotation	0

Use magenta for color and 2 for line weight.

You can see only block 2 and 5 are affected.

As we discussed before, this is because they use ByBlock properties. What we did was overridden block properties, so they follow new block properties regardless what layer they use. Now freeze Layer 1.



We can see block 4 and 5 disappeared because we created those objects on Layer 1.

Unfreeze layer 1 then freeze layer 2. Remember: we moved all blocks to layer 2.

What happen? All blocks disappear, includes block 4 and 5! So when we freeze layer 1, block 4 and 5 disappear. When we freeze layer 2, they also disappear!

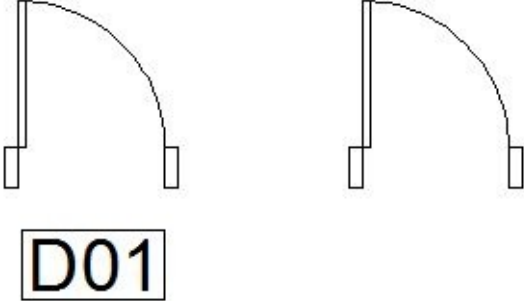
The blocks work as a container. All of the containers are on layer 2. Hiding layer 2 hides the blocks and all objects inside it. Doesn't matter on what layer the objects inside our blocks.

Unfreeze layer 2. Try to change properties and layers until you understand how they work with blocks. You may want to examine this behavior further by adding more objects on different layers on these blocks.

More layers practices

There are many benefits we can get by using layers. I mentioned that we could hide details for the column for example.

Another common use for layers in the block is to hide/show block tags. This door below has a tag on



another layer, so you can hide/show it when you need to.

ANNOTATIVE BLOCK

One of my favorite features in AutoCAD is annotation scale. Let's cover what it has to do with AutoCAD block.

We can use a block as an annotation. For example, [view title and room tag we created before is an annotation](#).

This block provides information with symbol and text. Our challenge in using annotation is we have to keep the annotation readable. When we draw on model space, annotation size for 1:100 obviously is different for 1:200.

Using layout/sheet gives us many benefits. At least there are [10 benefits of using layout](#). However, there are also downsides. A common practice when we use layout is to draw everything in model space, but place annotations in the layout.

1. When we need to show the same drawing in different viewports, then we need to draw several tags. When the design change, we need to change those tags. This process is tedious work and prone to mistakes. If we use annotation scale, we only need to update one.
2. When you need to move your viewport or pan your drawing within the viewport, your annotations does not move. You need to move the annotations manually.

Placing your annotation on model space eliminates these disadvantages. To have them appear with consistent size when we plot, we need to use annotation scale.

We do not cover annotation scale in this e-book. If you are interested in learning more about annotation scale, you can find more details here:

1. [Introduction to annotation scale](#).
2. [Controlling annotation scale further](#).

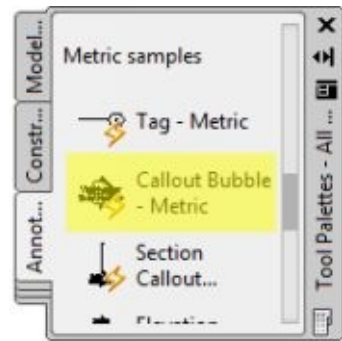
We are focusing on the block in this e-book, so we limit the tutorial to annotative block.

Understanding annotative block

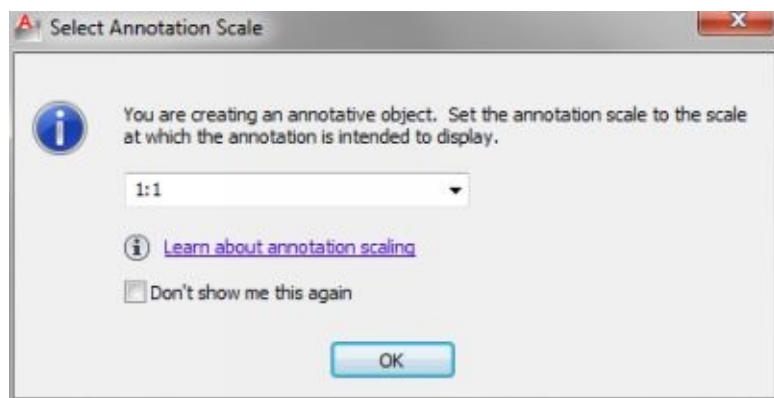
AutoCAD provides some annotative block samples. Let's try then to understand how they work.

Open your tool palette if it's not opened yet. You can open it by pressing [ctrl] + 3.

In tool palettes, find annotation tab. Find 'callout bubble – metric'.

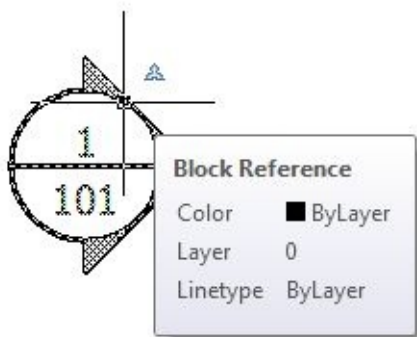


After you click it, you should see this dialog. Unless you already use it before and click don't show me this again. Click OK and accept the scale.



Insert the callout bubble to your drawing. It looks like a regular block, isn't it?

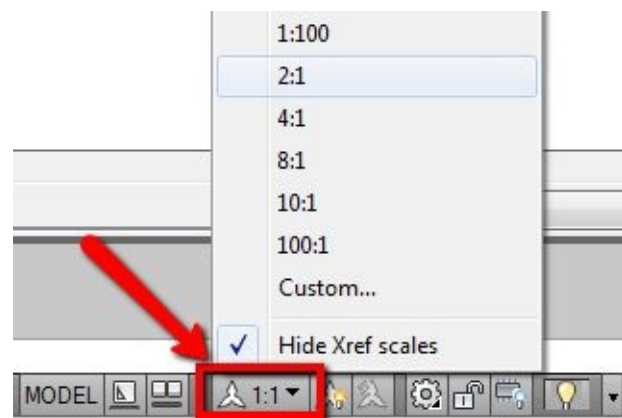
Hover your mouse pointer over it. The annotative block shows annotative symbol when your pointer is above an annotative block.



Try to hover your pointer above a regular block. See that there is no annotative symbol appears.

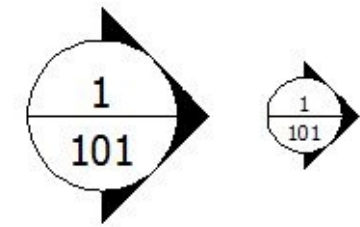
Examine AutoCAD status bar. By default, you should see AutoCAD is using 1:1 scale unless you changed active scale in the [previous dialog box](#).

Change active scale to 2:1. You can change it from annotation scale list button on the right side of AutoCAD status bar.



Insert the same block again. You may do it using insert command and choose the block from the list, or click the same block from the tool palette. The results will be the same using both methods.

What do we get? The last block is a half size of the previous block!



Try to change it to 1:2 then try to insert block again. That is how annotative block works. AutoCAD adjusts its size to fit current scale we are using.

Learn more about the annotative scale so you can work with it comfortably.

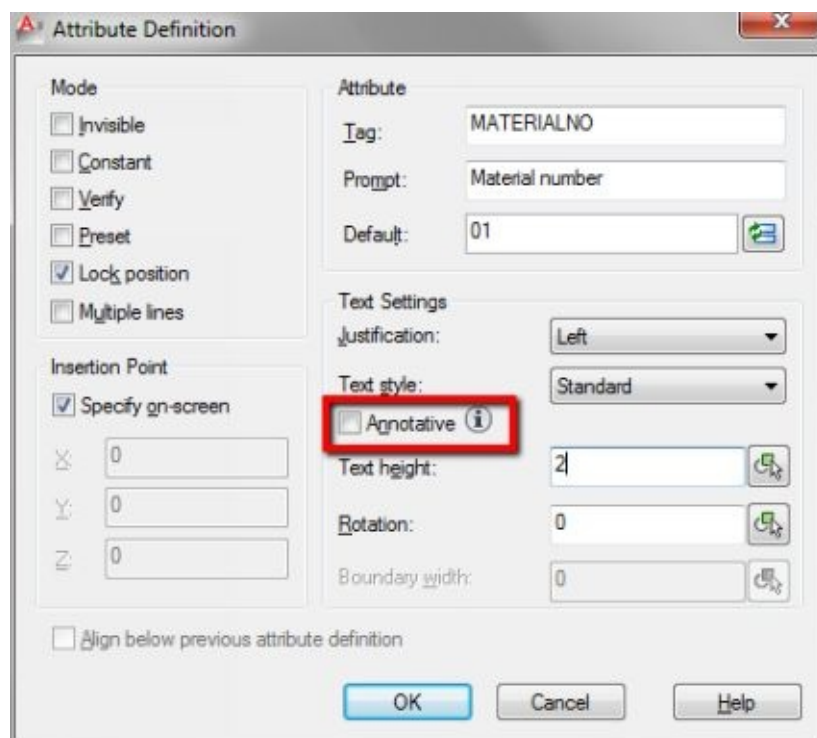
Creating annotative block

The idea of the annotative block is simple. When you use a certain scale, AutoCAD reads it and adjust block size as necessary. It makes your block has a consistent size in every plot scale you use. If you set the text to 2 mm height, then doesn't matter what scale you are using, AutoCAD plots it with 2 mm height.

Creating annotative block is similar to creating a regular block. Let's try to create one.

I recommend you to work at full scale when creating an annotative block. Draw it with plot size, and set the current scale to 1:1. If you want your text to plot at 2mm height, then place it using 2mm height.

1. Make sure we set the current scale to 1:1.
2. After you finished, create an attribute definition.
3. Fill tag, prompt and default value.

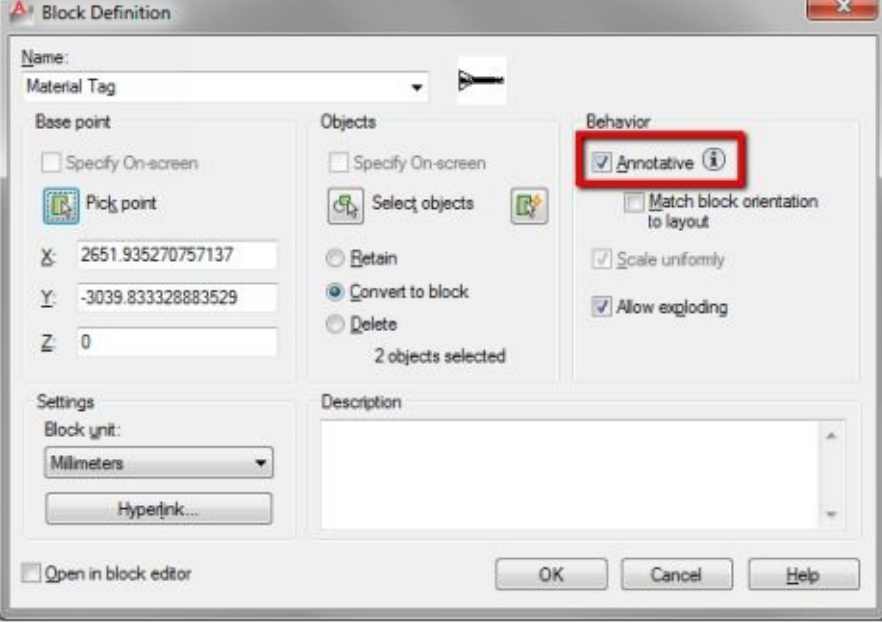


You do not have to check Annotative option in this dialog box. If you do, when you create an annotative block, AutoCAD sets it to not annotative automatically.

Finish your tag like below. You can use lines, polylines or polygon.



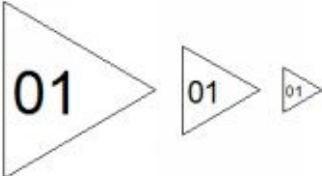
Create your block. The only difference between creating a regular block and an annotative block is you have to check Annotative option.



Click OK to define your block.

Using annotative block

Let's try to use this block. The one we convert before is already placed in 1:1 scale. Change current scale and insert your block.



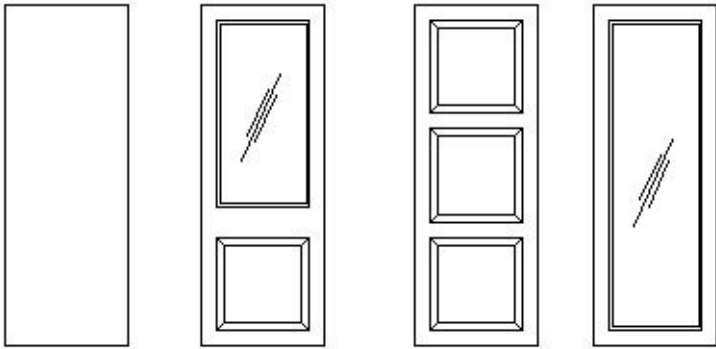
Remember: change active scale first, then insert your block.

ADDING INTELLIGENCE TO YOUR BLOCK

There are two ways you can add intelligence to your block. You can use actions or constraints. Autodesk introduces 2D parametric design or constraints in AutoCAD 2010.

Dynamic Block is a very powerful feature. We can add flexibility to our blocks. Until this chapter, we use blocks for typical symbols. When we need a typical symbol with minor differences, we have to create another block.

With dynamic block, we can manage similar blocks in a single block definition. Let's take these blocks as an example.



The blocks above are different door types. If we use regular block, we need to create four different blocks. With dynamic block, we only need to create one.

We can create a dynamic block in block editor. First, we need to create a regular block, then modify it.

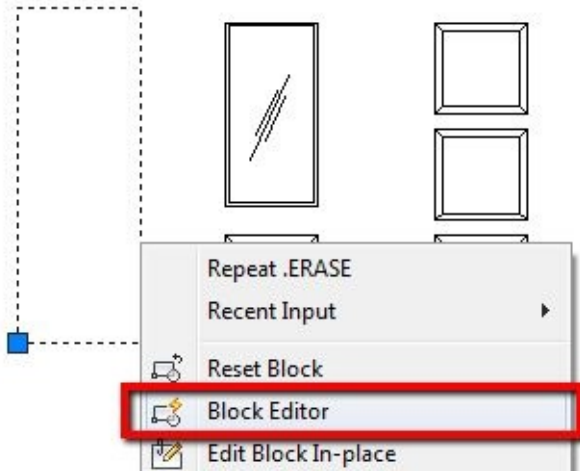
Using visibility states

Visibility state is the most basic functionality in a dynamic block. We simply define which objects are shown in different states.

Open file door-elevation.dwg. This file is included with this book.

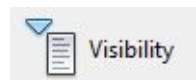
This file contains four blocks. They are door panel and three patterns for the door.

Select door panel block then right-click your mouse. Click block editor from the context menu.



AutoCAD opens this block in block editor.

To manage dynamic block visibility, we need to add visibility parameter. Find it in parameters tab, block authoring palettes.



Alternatively, you can click the small arrow below parameters group. Find it in *Ribbon > Block Editor tab > Action Parameters* tab. Click the small arrow to expand the list. If you think authoring palette takes too much of your desktop real estate, you can access actions and parameters here.



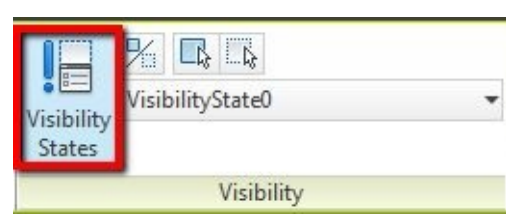
Click visibility parameter and place it at the right bottom of your door. Press [enter] to accept 1 as default number of grips.

You may place it anywhere you want it to. However, make sure you can find it easily. I recommend you to place it at the right bottom of your door.



This parameter shows warning sign. It means this parameter is not finished yet. You have some tasks to make it works.

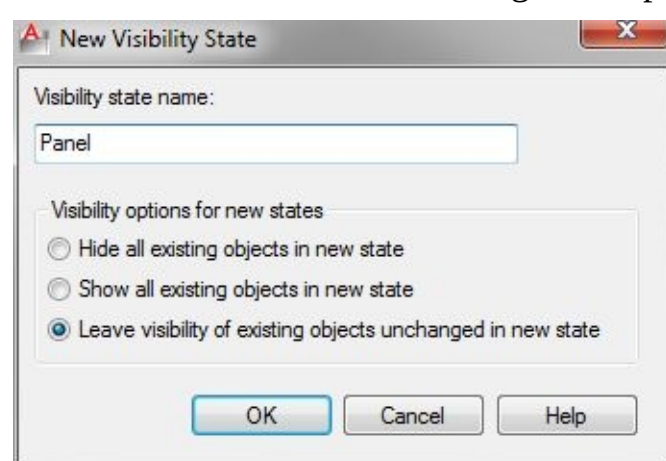
Click visibility states button on your *Ribbon* > *Visibility* panel.



By default, we have one visibility state named *VisibilityState0*. Select it and click *Rename* button. Change its name to *simple*. We are going to use it for *simple panel door*.

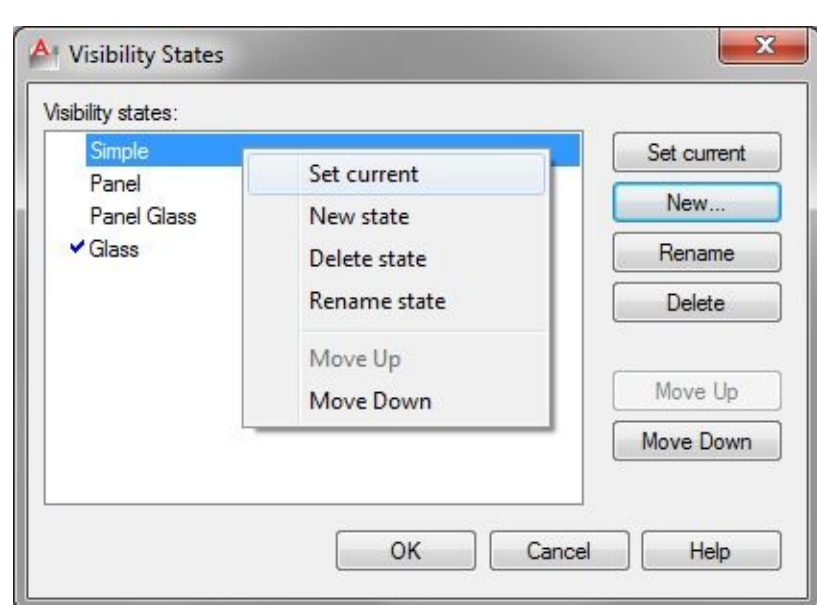
You can rename visibility state by clicking the name once, click *Rename* button, or right click then choose *rename state* from the contextual menu. Similar like renaming file name in Windows Explorer.

Create three new visibility states. Give them name *panel*, *panel glass*, and *glass*. When you create a new state, leave it to the default setting. Just input the state name.



After you finish, select *simple* and click *set to current*. AutoCAD sets current visibility state to the last one you created. We start working on *Simple* state, so set it as current.

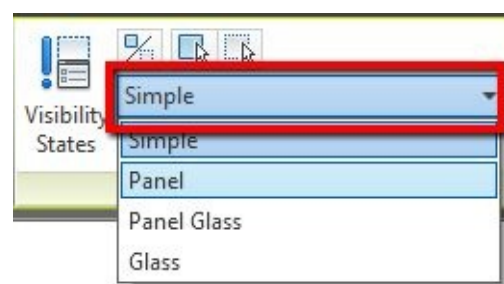
You should see the check mark next to the state name.



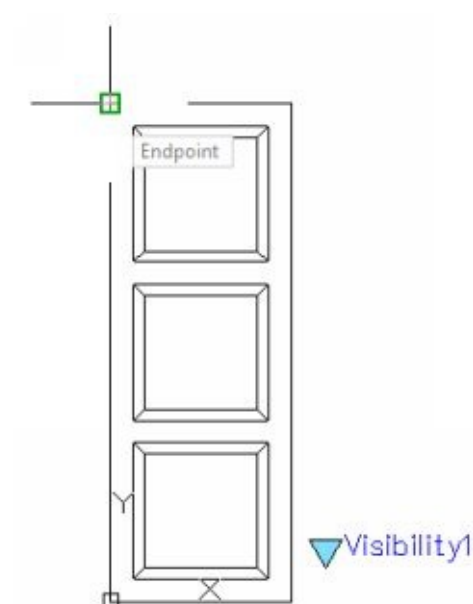
Click OK.

The warning sign disappears. However, it does not mean we are done. What you see now is a simple door. We saved it in the *Simple* state. Now we are going to set the other states.

Change current visibility state to the *Panel*. You can do it by switching from visibility panel in block editor contextual ribbon.



Activate insert block, select panel block from the list. Use the top left corner of the door as the insertion point.



We have defined how our door looks in panel state. You may draw more objects to define how your door panel will look like.

Switch to the *Simple* state and back to panel state. Have you seen the difference?

Now change current state to panel glass. You should see the door revert to the empty simple door. Insert panel and glass block at the same insertion point.

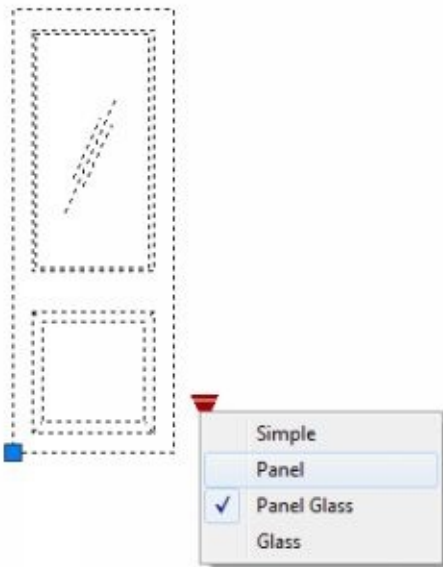
Do the same steps for the last block. Change visibility state to the *Glass* state and add the glass block.

You have defined all four visibility states. The door should look like in different states. Test it by switching between all state from the ribbon.

Save your block and close block editor.

Test your block. Select single door block. It is the left most of your doors. Remember, we add visibility states and other blocks to this block. You may delete other blocks to avoid confusion.

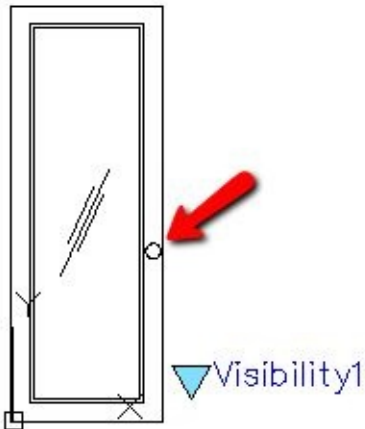
You should see visibility control at the bottom right of your door, or wherever you put it in the making process. Click it and change the visibility state.



Adding objects to current state

Open the block in block editor again. Set glass as the current state.

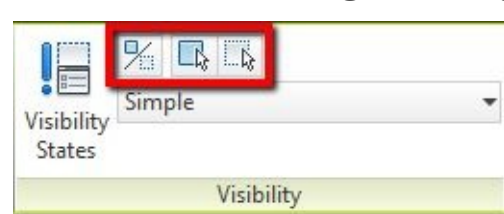
Draw a circle as a door handle. Don't worry about its size and location. It is not critical.



Change to the *Simple* state. The door handle disappears. It is because we draw it on glass state, so it only appears when we change current state to glass. This concept is similar to drawing on layers.

We want to make it appears in every state, so we have to adjust its property.

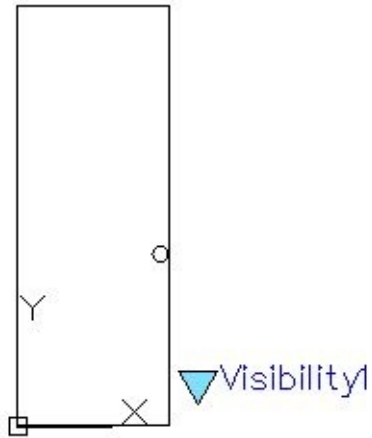
Controlling visibility states



We can control visibility using these three buttons.

Click the left most button. It is visibility mode button. It shows invisible objects in this block. That includes objects that are in other states.

Click the second button. It is *Make Visible* button. If the visibility mode is off, your hidden objects will be temporarily shown here. Select the circle and press [enter] to end selection. Change visibility mode again.



Now we can see the door handle in this state.

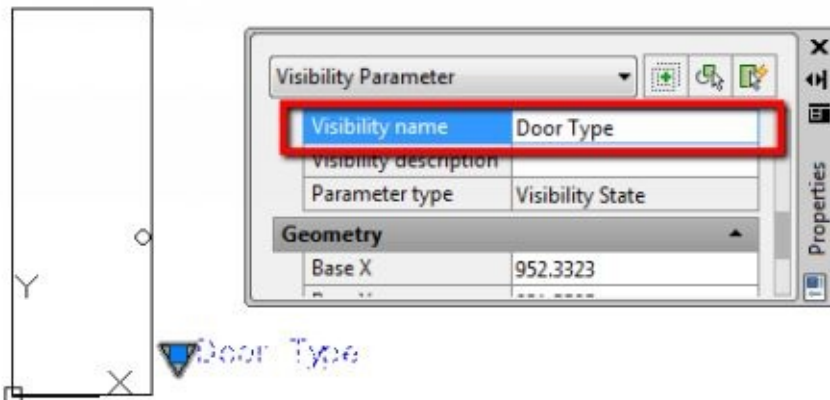
Change current state to another state. Repeat the same process to make the handle appears in every state.

Changing parameter name

This block works as expected. However, working with a complex dynamic block that has many parameters can be confusing. Instead of using default parameter name, it would be easier for us to recognize what does the parameter do. I recommend you to rename your parameters because we can also use it in data extraction.

[Data extraction will be discussed later in this book.](#)

We are still working in block editor. Select visibility parameter and change parameter name from properties palette. In this example, I use door type.



Using parameters and actions

Actions simulate what you do in AutoCAD. To modify objects in AutoCAD, we move, array, stretch, or rotate our objects. These actions are what we use to build a dynamic block.

To make those actions work properly, we have to relate actions to parameters.

Let's take an example.

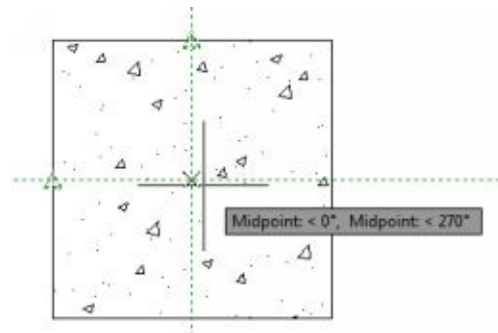
If you want to rotate objects, in the dynamic block we use rotate action. AutoCAD knows we want to rotate the objects. However, AutoCAD needs more details.

To rotate objects, we need to define base point and angle. To do this, we use rotation parameter.

Adding parameters

Let's create a simple block. We are going to create a concrete column. Create a rectangle 500x500mm. Add concrete hatch inside the rectangle. Make sure this hatch is associative.

Create a block from the drawing, make the column center as the base point.



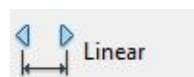
After you finish, open it in block editor.

Rectangular columns in design have different sizes. Instead of creating each size in separate blocks, we make one block and add the ability to resize it in our drawing.

How do we change the size of the column? We use the stretch command. Stretch allows us to change the column size along the horizontal axis and vertical axis.

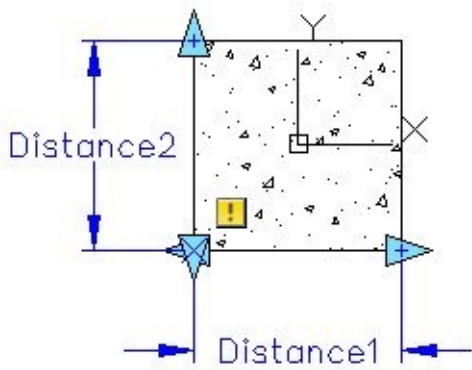
Now we know what action we can use. What about parameter? Which parameter should we use?

Stretch works with a linear parameter. Activate linear parameter.



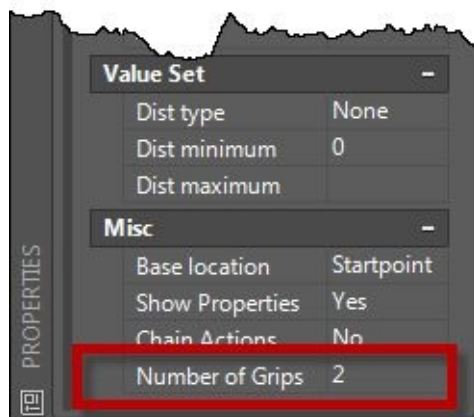
Let's add the parameter to our rectangle sides. Snap to its corners to define which side we want to stretch.

When you place the parameter, click bottom left point first. The first click defines base point for your linear parameters.



As before, we can see the warning sign. We still have things to do.

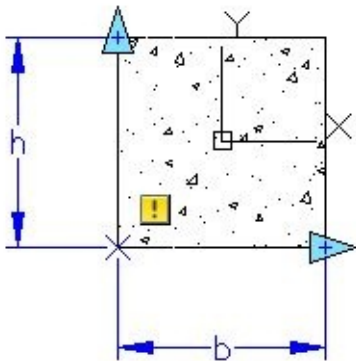
Before we continue, open properties palette. Select distance1 parameter, and change the parameter name to b. Modify the number of grips to 1.



Do the same with distance2. Change parameter name to h, and the number of grips to 1.

We reduce the grips number because we only need to stretch it to one direction.

If you do not click bottom left point first, then you may see the arrow disappear. This option will remove the 2nd arrow that you defined when placing parameter.



You can also remove unnecessary arrow by selecting them and press delete.



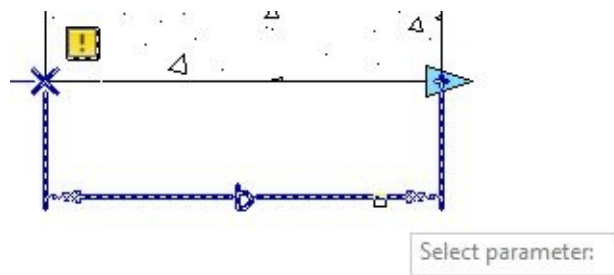
Link action to parameter

We placed parameters. Now we can add action and link it to our parameter.

Activate stretch action.

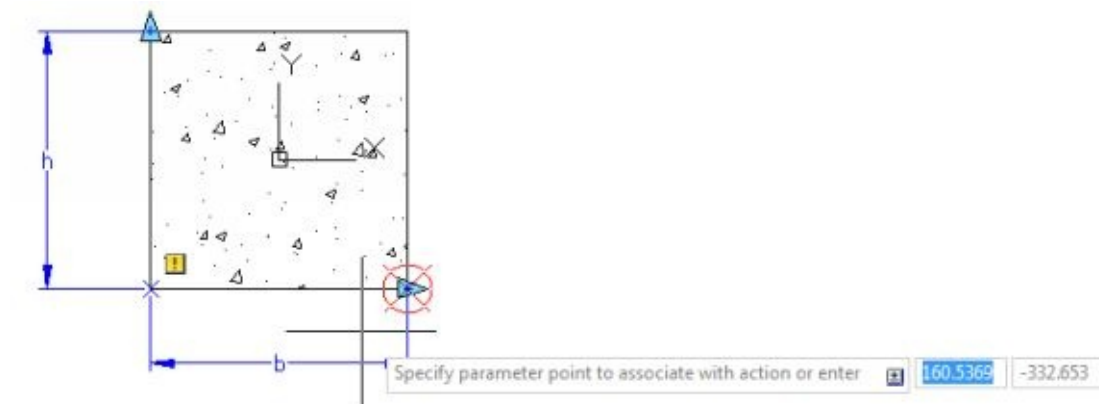


AutoCAD asks you to select a parameter. Select b parameter.

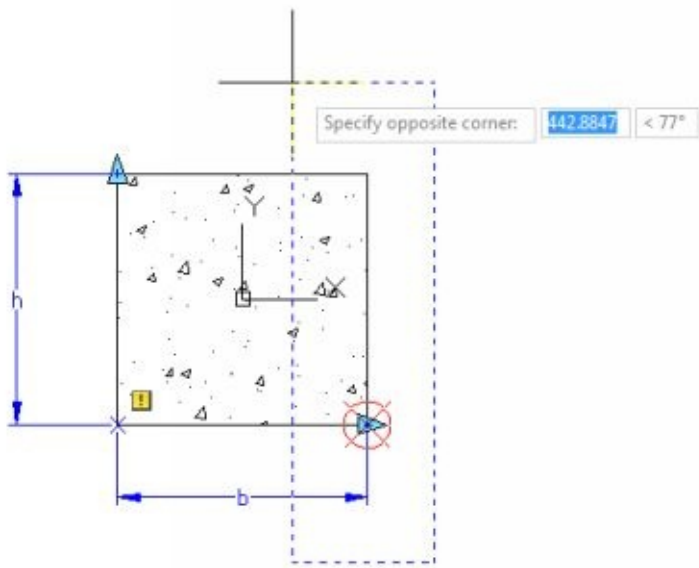


Now AutoCAD knows which parameter to link with this action. Now we need to define which point to associate with this action. Move your pointer to the right side of this column.

You should see red circle appears at the bottom right corner of our column. Click your mouse.



Next, we need to define our stretch frame. Defining stretch frame is similar to defining stretch frame when you stretch when modifying objects. Use image below as a reference.

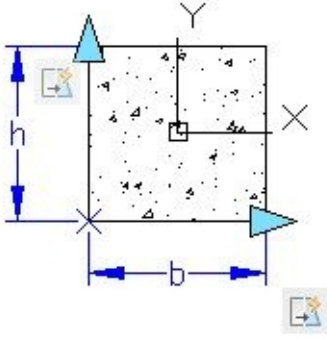


Select the rectangle and press [enter].

We are done with the first action.

Repeat the same process to add a stretch action for h parameter.

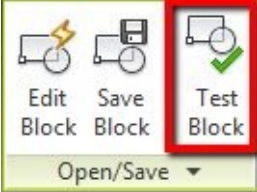
Our finished block should look like below. You should see two stretch action icon near the arrows.



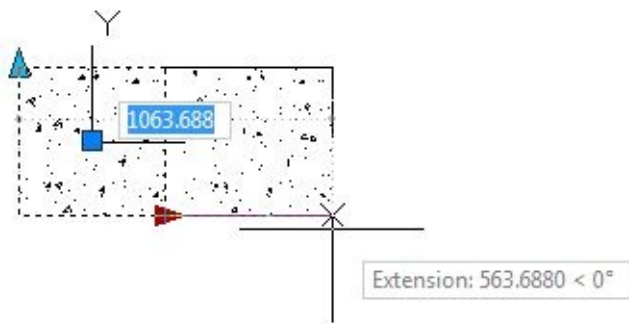
Testing block

Let's test this block before we continue. Testing the block regularly after add a few actions is a good practice. Testing the block allows us to find what is wrong and quickly fix it. If you test it after you add many actions, it can be hard to trace your mistakes.

Click test block in AutoCAD ribbon.

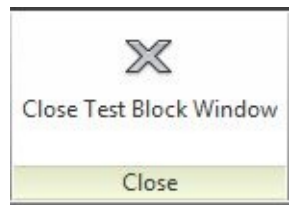


Now we are working in test block window. Click our block to select it. You can see arrows from actions we added before. Click the arrow and move your mouse away. This action resizes the column.



If your stretch actions do not work as you expected, you need to check your parameters and actions to see what's wrong.

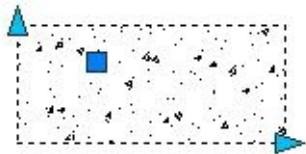
Click close test block window.



Parameter properties

Now our stretch actions work. However, if you notice, it does not work as we need it.

After we stretch it, our action shifts our insertion point. If we want to keep our base point at the center, we have to stretch it using the same value to opposite direction. This is not really what we want it to work, isn't it?



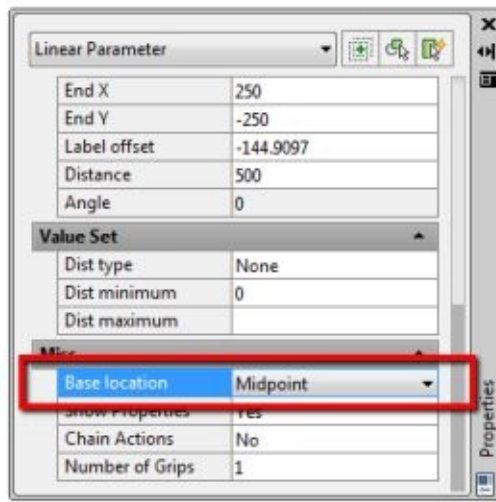
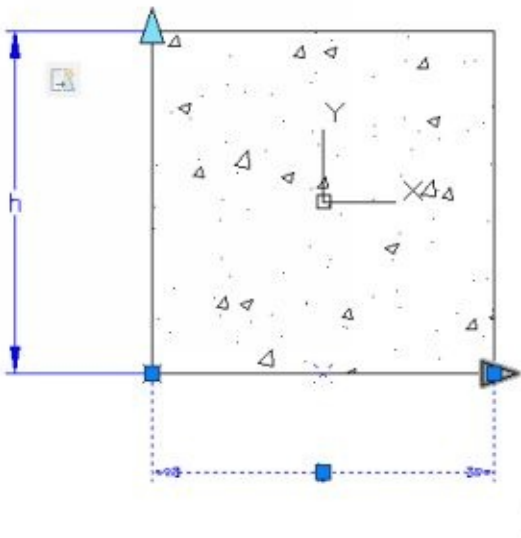
The other problem is we, and other users can stretch it to any size we want. This behavior is not right. We want this column to have a precise size. Moreover, we might want to limit its minimum and maximum size.

We are still working in block editor. If you already close block editor, select your block, right-click, then select Block Editor from the context menu.

Linking stretch action to opposite direction

Open properties palette then select linear parameter.

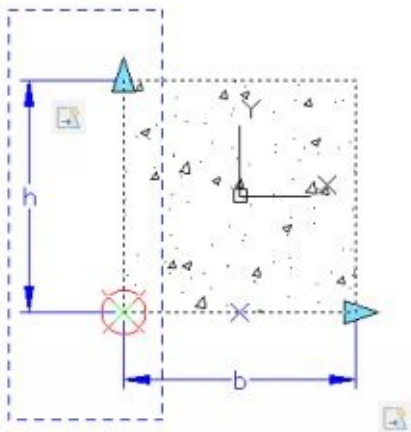
In the Properties palette, change the base location to the midpoint.



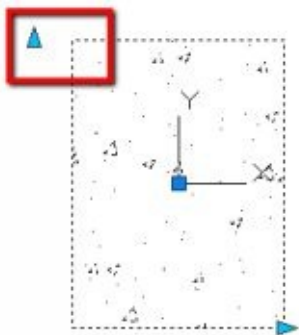
By changing the base location to the midpoint, our parameter stretch in both directions just by using single action. This behavior can keep our insertion point at block center.

The parameter stretches to both sides now, but we have to add one more action to stretch our object to the left.

Add action like we did before, using b parameter.

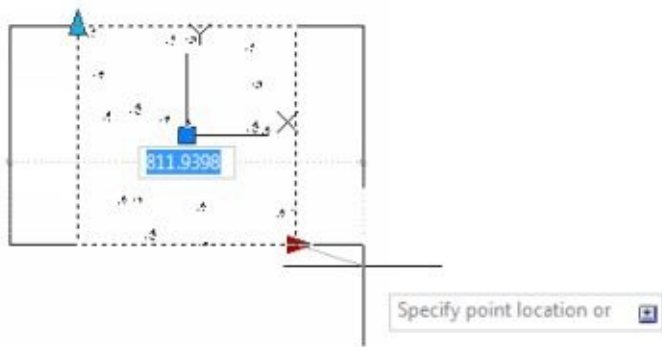


This time you have to select the grip in object selection. Select the grip to make sure the arrow stays at the rectangle corner. If we do not include it, it still works properly. However, the arrow is not at the right position after we stretch it.



To avoid confusion, you can simply select all objects in this exercise.

Now if we test this block, if you stretch it, you can see it resizes to both directions.

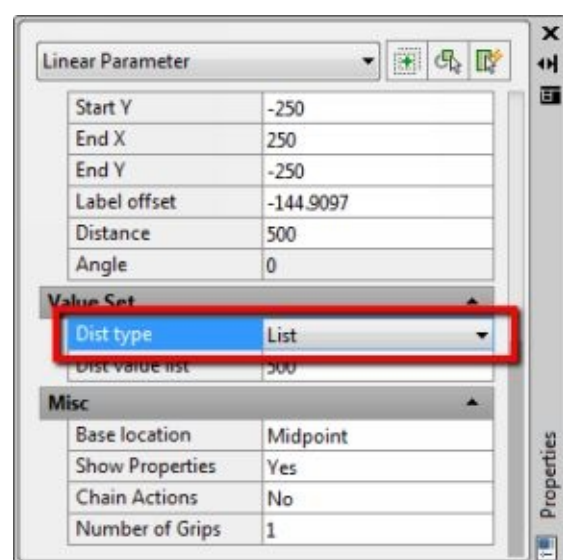


Repeat the process for h parameter and add stretch action to the top of this column.

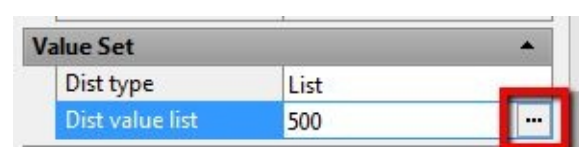
Limiting size using list

Now we are going to limit the column size. Select b parameter again.

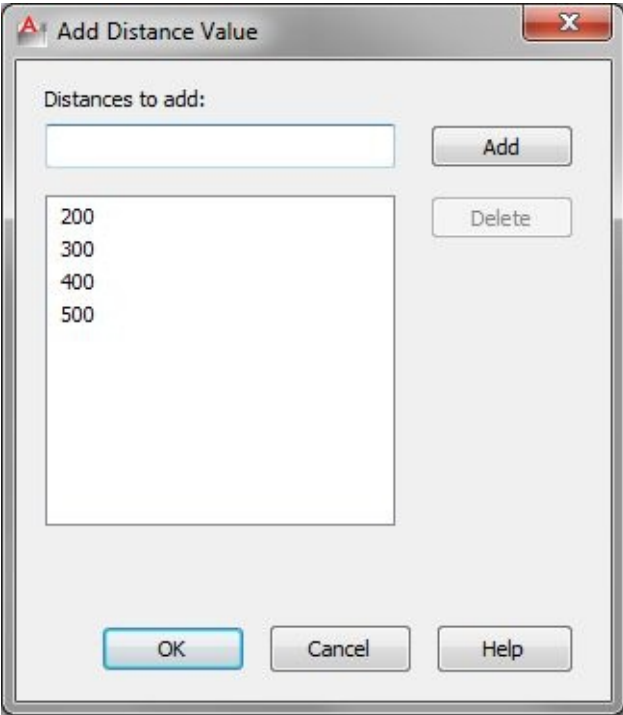
In Properties palette, change *dist type* to *list*. By defining the distance type to *list*, we can limit the users only can select predefined distance.



After you change *Dist type*, click ... button on the right side of **dist value list** field. You need to click the field first before the button appears.



In opened dialog box, type distance you want to add to the list and press enter. Alternatively, you can click Add after each entry.

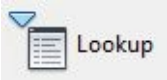


Repeat the same process to h parameter. Don't forget to test your block after you finish. You may add other sizes that you consider appropriate.

Create a list of column size

By defining a list, we allow users to stretch column only to specific sizes. However, if they need to change both directions, they must stretch it twice. We can expand this capability further. Let's edit the block again in block editor.

Insert lookup parameter to your block.

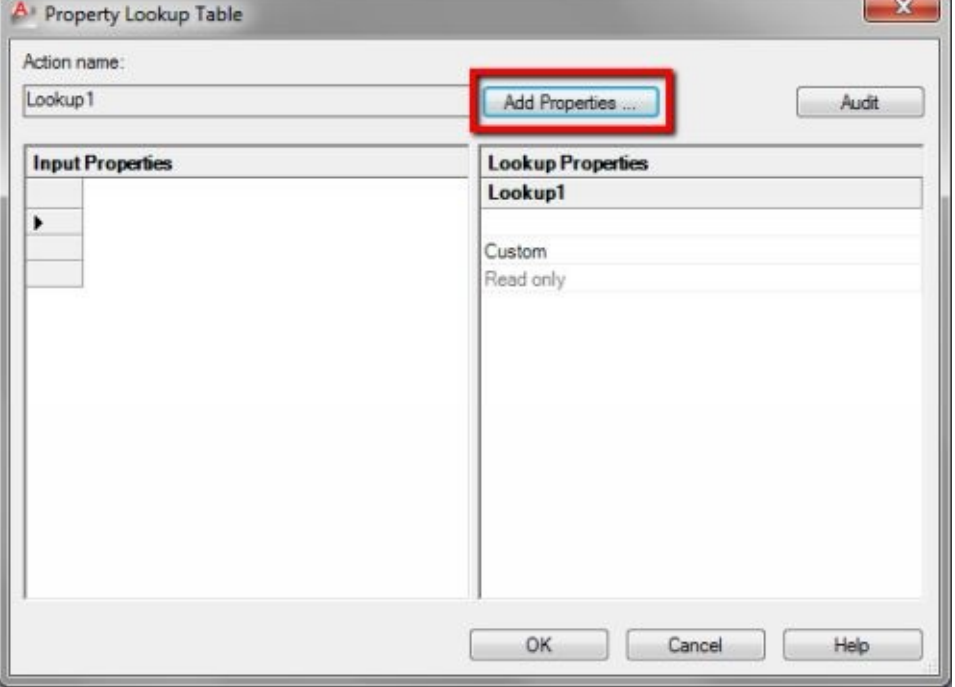


Again, we see the warning sign. We need to add lookup action. Insert it from your authoring palette.



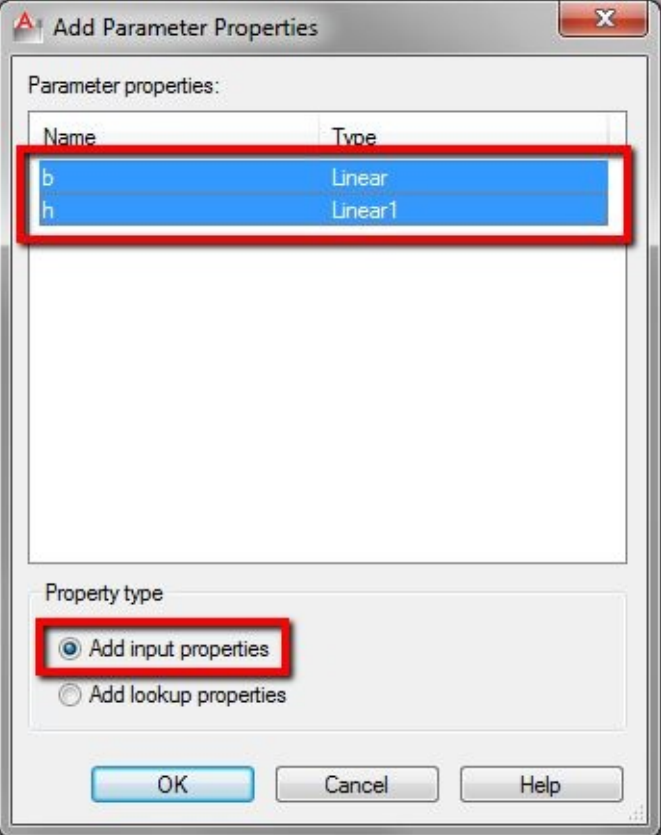
After you place lookup action, select the parameter. AutoCAD opens a dialog box like below.

Click add properties to associate this action to our linear parameters.



Another dialog box opens. You should see b and h parameters here. Hold [ctrl] and click both of them. We are going to use them all.

Make sure you select *add input properties* in property type.



Click OK.

Now we can set what types and sizes we want for our column.

Below lookup properties, fill your column types. You can type C1, C2, C3... for this exercise.

Lookup Properties	
Lookup1	
C1	
C2	
Custom	
Read only	

Click on the same row below b parameter and choose the C1 size. Do not forget to choose h value.

Input Properties		
	b	h
▶	300	<input type="text" value="300"/>
		200
		250
	<Unmatched>	300
		400
		500

Set every column size as you many as need.

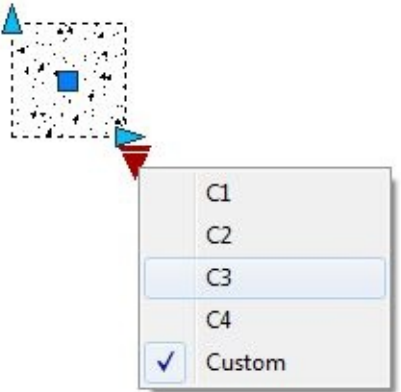
Input Properties			Lookup Properties	
	b	h	Lookup1	
	200	200	C1	
	200	300	C2	
	300	300	C3	
▶	300	500	C4	
			Custom	
	<Unmatched>		Read only	

After you finish, click OK.

You may want to change the parameter name from lookup1 to something that makes more sense.

Property Labels	
Lookup name	Column Types
Lookup description...	
Parameter type	Lookup

Now save your block and test it.



Instead of stretching the column, you can choose it from the list. Using a list prevents users from placing

nonstandard column size, and of course, it is faster than stretching it twice.

CONTROLLING BLOCK WITH CONSTRAINTS

In the previous chapter, you learned how to create a dynamic block. Since AutoCAD 2010, Autodesk introduced the parametric capability in AutoCAD. We can use constraints to give our blocks intelligence like parameters and action we did before.

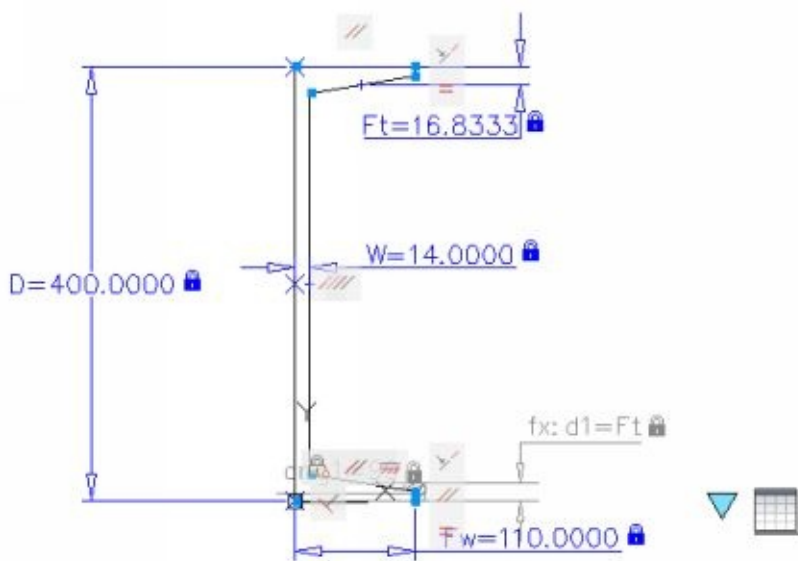
Parameters and actions in dynamic blocks allow you to modify block like when you do it in drawings. You can stretch, move, or rotate your objects within a block.

Constraints in parametric work differently. Constraints give you the ability to control the geometry and object size. Parametric can be an option if you need to define size and relationship between geometry.

There are some good examples in AutoCAD tool palettes. Let's try a channel beam.



The way we can change the beam profile is different between parameters and actions.



Here, we define how an object relate to the other. Should it be parallel to adjacent line? What's the line length? Is the length relate to other objects length?

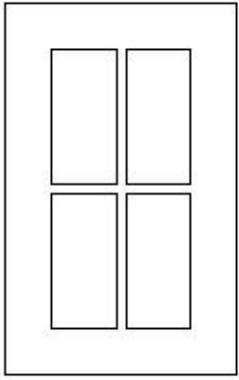
Parametric design has been used in the manufacturing industry for decades. Imagine if you have a standard part. It has the same geometry but has different sizes. It is a lot more complex than scaling that part!

We do not cover parametric design in details in this e-book. If you want to see about its capabilities, try these resources:

1. [AutoCAD parametric design on YouTube by DSCOHN.](#)
2. [AutoCAD geometric constraints part 1 on Daily AutoCAD.](#)
3. [AutoCAD geometric constraints part 2 on Daily AutoCAD.](#)
4. [AutoCAD dimensional constraints part 1 on Daily AutoCAD.](#)
5. [AutoCAD dimensional constraints part 2 on Daily AutoCAD.](#)

Using dimensional constraint

Let's do an exercise. Open window.dwg.

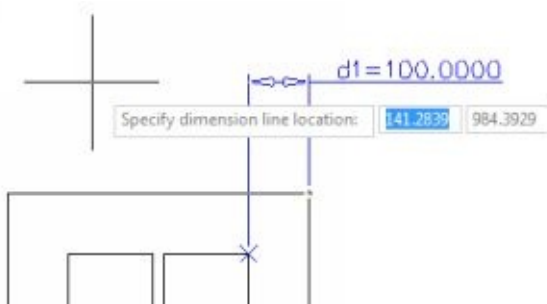


This drawing has one window block. We will add dimensional constraints so we can control the window size. The challenge here is we need to control glass panel, and windows frame as well. Using stretch actions are not a good option here. It may require many actions and can be very complicated.

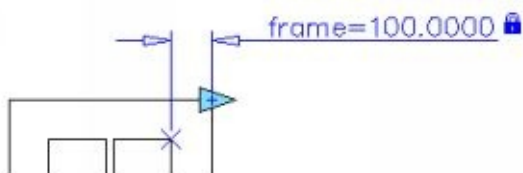
Select the window block, right click and choose block editor from the contextual menu.

In your contextual ribbon, dimensional panel, activate linear dimension.

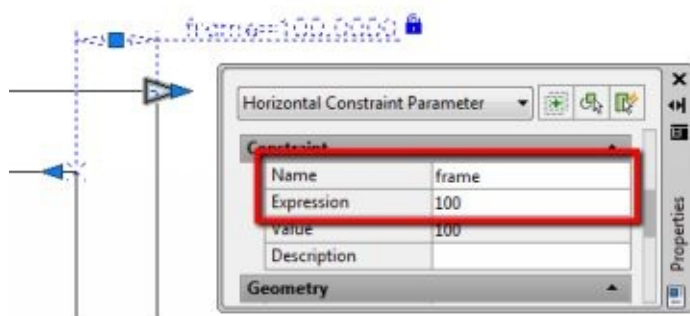
Add linear dimension like below.



After you click the points, you can type to change the constraint name and expression.

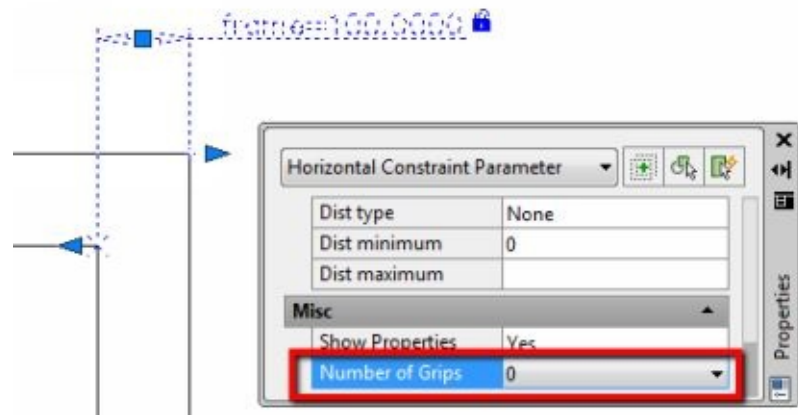


To accept default value now just press [enter]. You can modify the values later on properties palette.

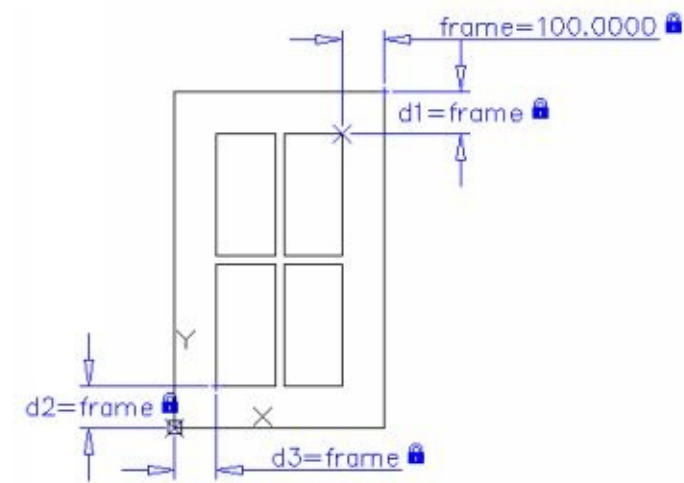


Similar to parameters in the dynamic block before, we can change our window size by dragging it. This time, we want to control the sizes completely using a table.

While you select the parameter, change number of grips in properties palette to 0.

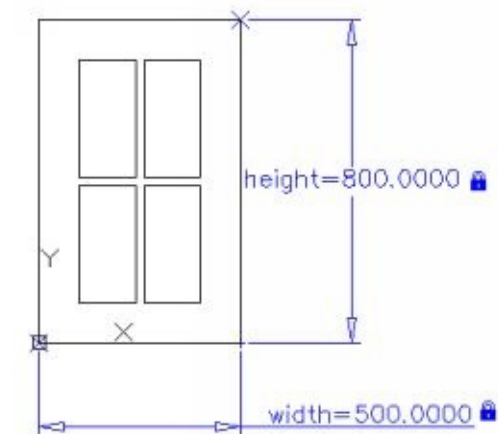


Add all constraints to control frame size as below. Don't forget to change the expression to 'frame' parameter. By changing the expression to *frame*, it links to frame parameter we placed before, and take its value.



Add two more dimensional constraints to define window height and width.

**Previous dimensional constraints are hidden for visibility purpose.*



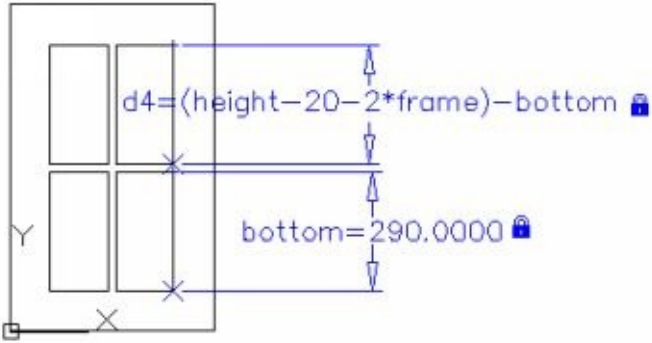
Add more constraints to define panel sizes. We can make it possible to have different size for top and bottom panel by using a simple formula.

We allow users to change bottom panel height, and top panel adjusts automatically. We set frame between panels to fix size: 20mm.

Then top panel formula should be:

$$=(\text{height}-20-(2*\text{frame}))-bottom$$

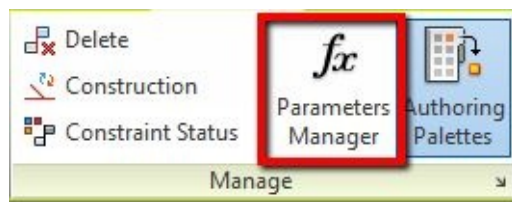
See image below as a reference.



We have set all dimensional constraints. Let's test it.

Changing parameters value

Activate parameters manager.



You can see another palette opened. Notice that every dimensional constraint we created before are listed here.

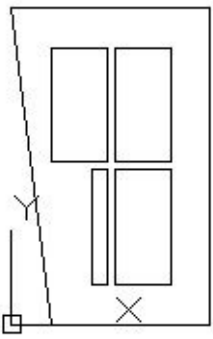
The Parameters Manager palette is open, showing a list of parameters. The list is titled "Dimensional Constraint Parameters" and includes the following entries:

Name	Expression	Value	Show
bottom	290	290	Yes
d1	frame	100	Yes
d2	frame	100	Yes
d3	frame	100	Yes
d4	(height-2...	290	Yes
frame	100	100	Yes
height	800	800	Yes
width	500	500	Yes

At the bottom of the palette, it says "All: 8 of 8 parameters displayed".

Try to change width value to 400. Remember, you need to change it under expression column.

You may see a different result than what I have below, but you should see the same mess as I do. Change any other parameter to see how it works.



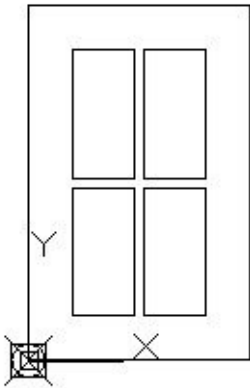
Dimensional constraints only move constraint points. If we only use dimensional constraint alone, it's hard to predict how it works.

To keep the window geometry, we have to use geometric constraints to tell AutoCAD how the objects relate to each other.

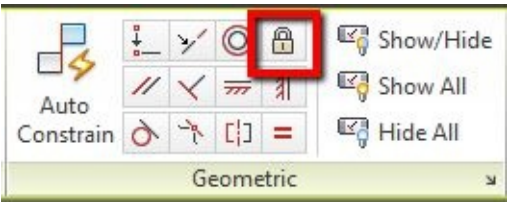
Click UNDO until you see the window to its original form before we change any parameter value.

Using geometric constraint

The first thing we have to do is lock a point as a fixed point. Insert a point using POINT command at 0,0.

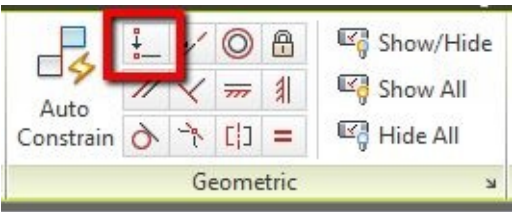


Now lock the point using fix constraint. Activate the tool and select the point.

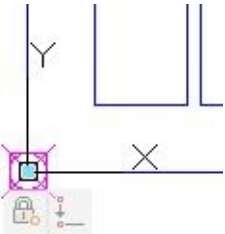


We use a fix object as a reference for other geometry.

Next, we want the windows corner point to always coincident with our reference point. Activate coincident constraint.



Click rectangle corner, then click the point. Now we have two geometry constraints in our block.

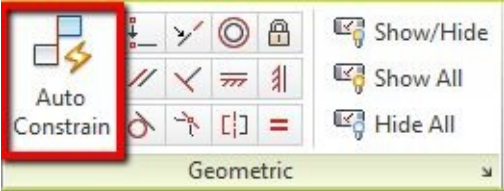


This constraint keeps our window at the reference point. We need it because we want full control of our base point.

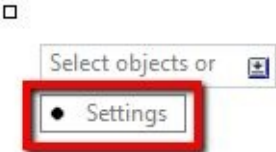
Using Auto Constrain

You can try to apply all geometry constraints manually to this window. However, we are not focusing on constraints, so let's just use auto constrain and let AutoCAD automatically complete it.

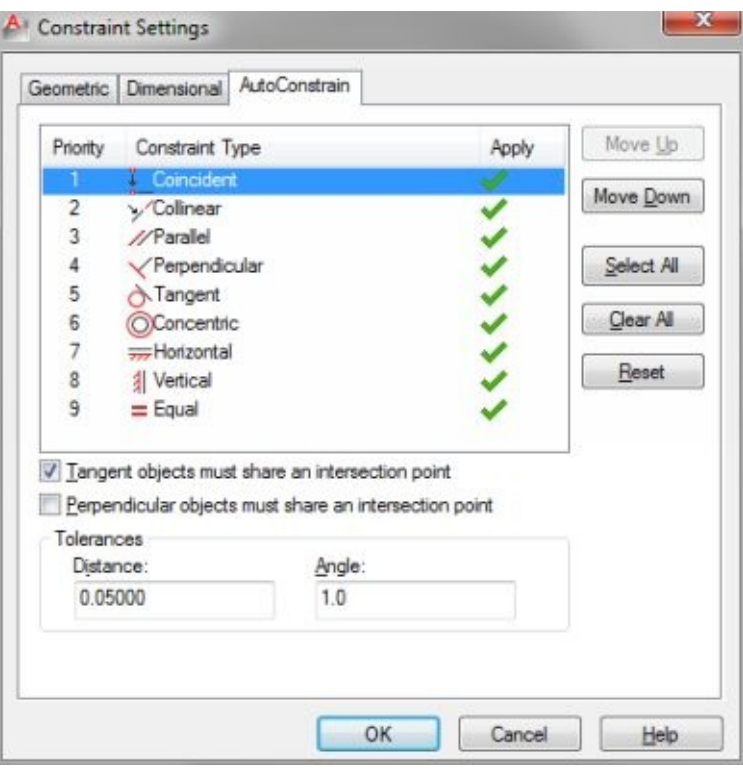
Activate auto constrain.



Activate Auto Constrain, type S then [enter] to change the settings. Alternatively, you can press down arrow to activate it using the Dynamic Input.



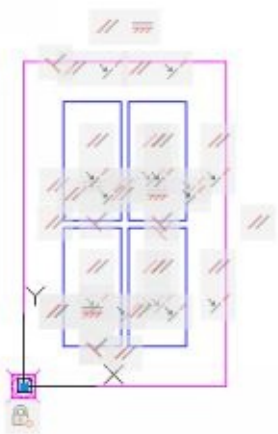
In Auto Constrain settings dialog box, activate all constraints.



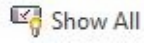
Click OK.

Auto Constrain is still active. Select all objects then press [enter] to end selection.

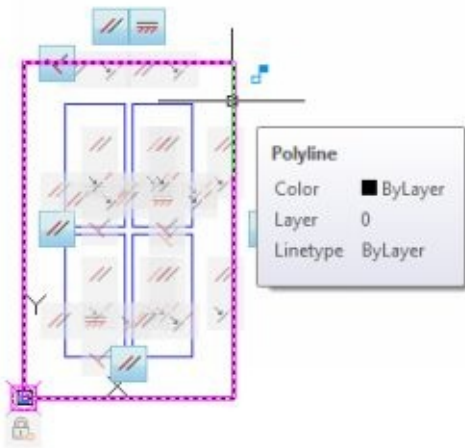
You should see all of the constraints applied by auto constrain.



If you do not see them, click show all in geometric constraints panel.



Hover your mouse pointer over an object. AutoCAD highlights all constraints that relate the object to other objects.

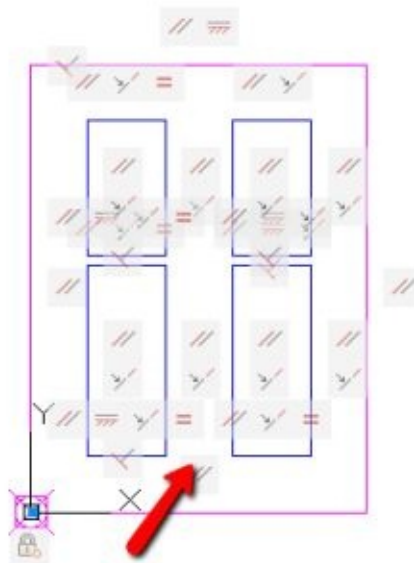
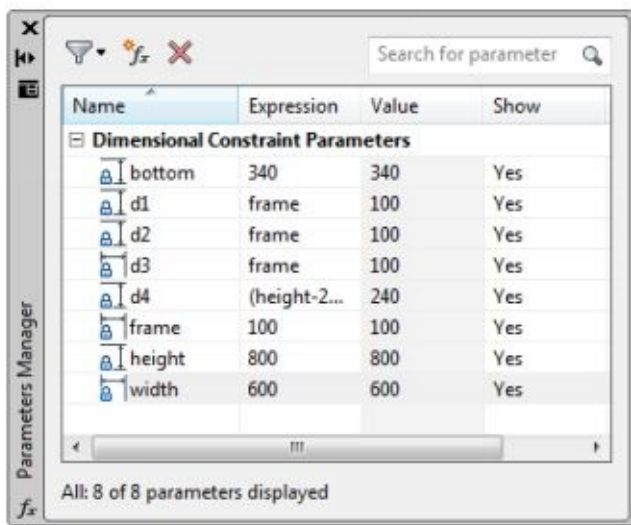


Let's test our window again. Change windows height, width, and panel size.

Do not change expressions that use formula or refer to other parameters.

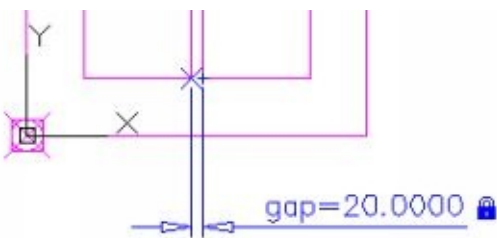
Now it is much better. However, there is one more thing to fix.

The gap distance between panels is still not correct. We can fix this by adding one more dimensional constraint.



Add linear dimension and type **gap=20** to change parameter name and value. Regardless the current gap was shown.

After you typed it, the geometry adjusts to the new value.



This window now should works.

Let's do a final modification to our parameter. In Parameters Manager Palette, find a parameter that defines our top panel height. Change 20 to *gap*, so we can control the size using the parameter.



d3	frame	100
d4	(height-gap-2*frame)-bottom	240
frame	100	100

Parametric blocks allow you to control objects by their sizes. Not by actions like what we do when drawing with AutoCAD. If you need to control objects by modifying them, then the dynamic block actions are more suitable.

Using block table

We can control object size using grips, but this time we use block table. It is very similar to the [lookup table](#); the only difference is block table is for the dimensional constraint.

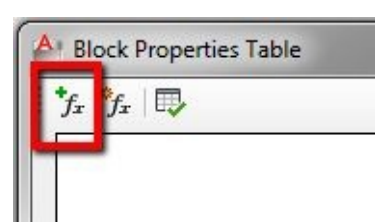
Activate block table.



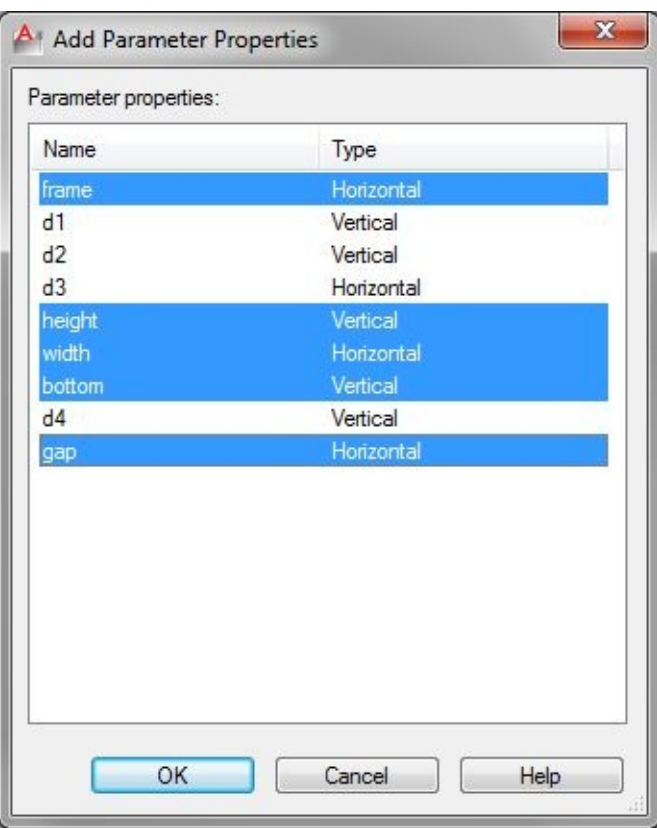
AutoCAD asks you to place the block table. Click somewhere below the base point.

Next, AutoCAD asks you to define the number of base point. Press [enter] to accept 1 as the default value.

AutoCAD opens a dialog box. Click add properties at the top right of this dialog box.



Another dialog box opened. Hold [ctrl] then click parameters name to select it. Select frame, height, width, bottom, and gap parameters.

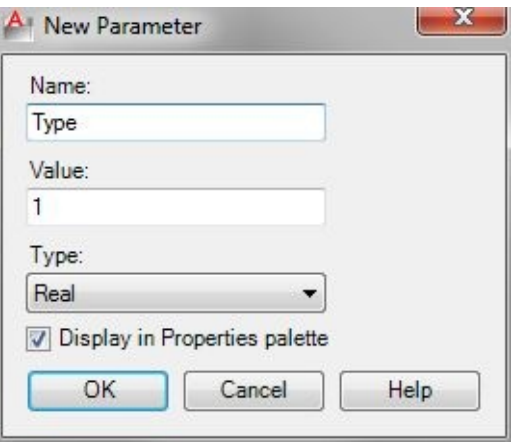


Click OK.

Now click **add user parameter** to define our parameter.



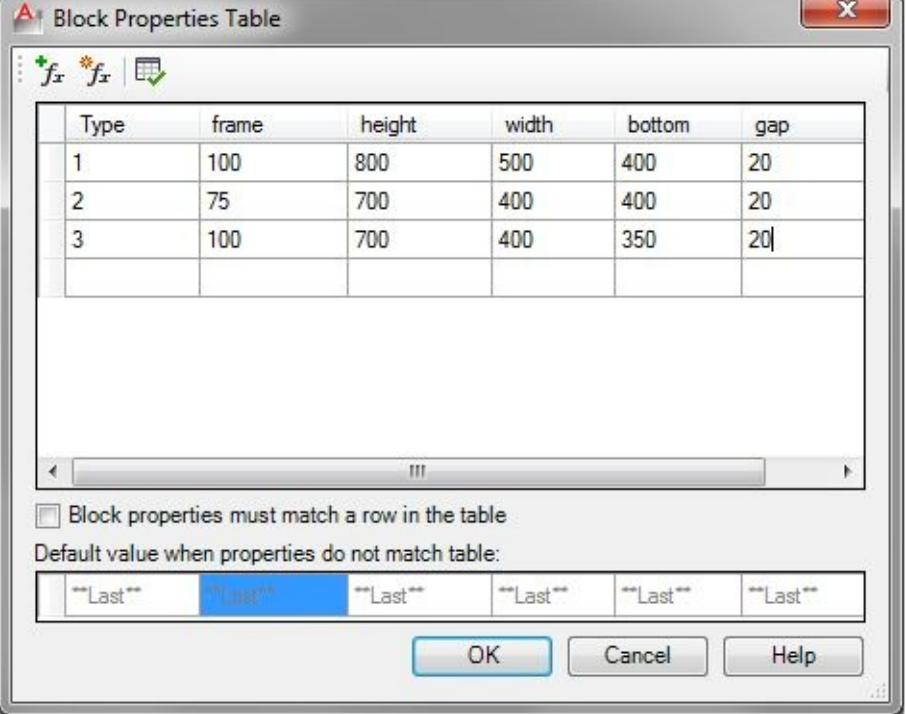
In opened dialog box, fill **type** for parameters name, and change this parameter type to real.



Click OK.

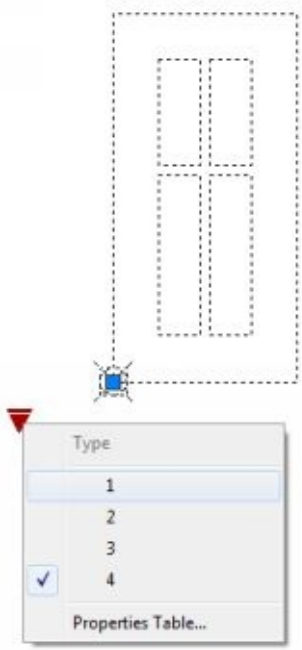
Now we can add values to our table. Click in the cell to type your values.

In this example, I use 3 windows data. You can add more as necessary, for your exercise.



Click OK, then save your block. Close block editor.

Test your block by selecting it and click the down arrow. You can select windows type and change its size now.



If you see trailing zero in this selection, it is because it follows your decimal unit precision.

More about dynamic blocks

This part is the end of our dynamic block basic tutorial. There are many possibilities that you can do with it. You can find more examples in a later chapter in this e-book.

EXTRACTING DATA

We have done many things with our blocks. You may find it useful. However, there are more things you can do. By using blocks, it is not only you can finish your drawings faster. You can also generate reports from them quickly.

In this chapter, we extract data from our blocks.

Information in your blocks

We learned that we could add information to our blocks. How can we use it? We can extract the information and report them in a table.

There are data we can get from blocks. You need to know these data types so you can plan which information you should add to your blocks.

Attributes

We can add as many attributes as we want. If we have a door block, AutoCAD only recognize its name. You can add more data such as manufacturer, fire rating, door accessories, and any other data that you consider important. We have done this [earlier in this e-book](#).

Parameters

We can use parameters as information as well. In column example, we can stretch our column so we can have different sizes. We have b and h parameters to show the column size. Moreover, we also have column type from a lookup table.

We can extract these parameters to our table.

Using fields in Attributes

Fields are an extremely useful feature for automation. A sample of using fields is to put common data in title blocks. When your drawing information changed, the information can be automatically updated. We can use it for plot date, last saved, and other common information.

We can see more examples of fields in block samples later.

Using data extraction

Open data extraction.dwg. This building plan was created using blocks. Walls, doors, and windows are typical building elements. Therefore, we can consider to create them as a block and reuse them in our design process.

There are nothing special about the blocks. You should be able to create them after reading previous tutorials. You might want to examine them before you start using data extraction.

Let's activate data extraction. It is on Annotate tab, Tables panel.

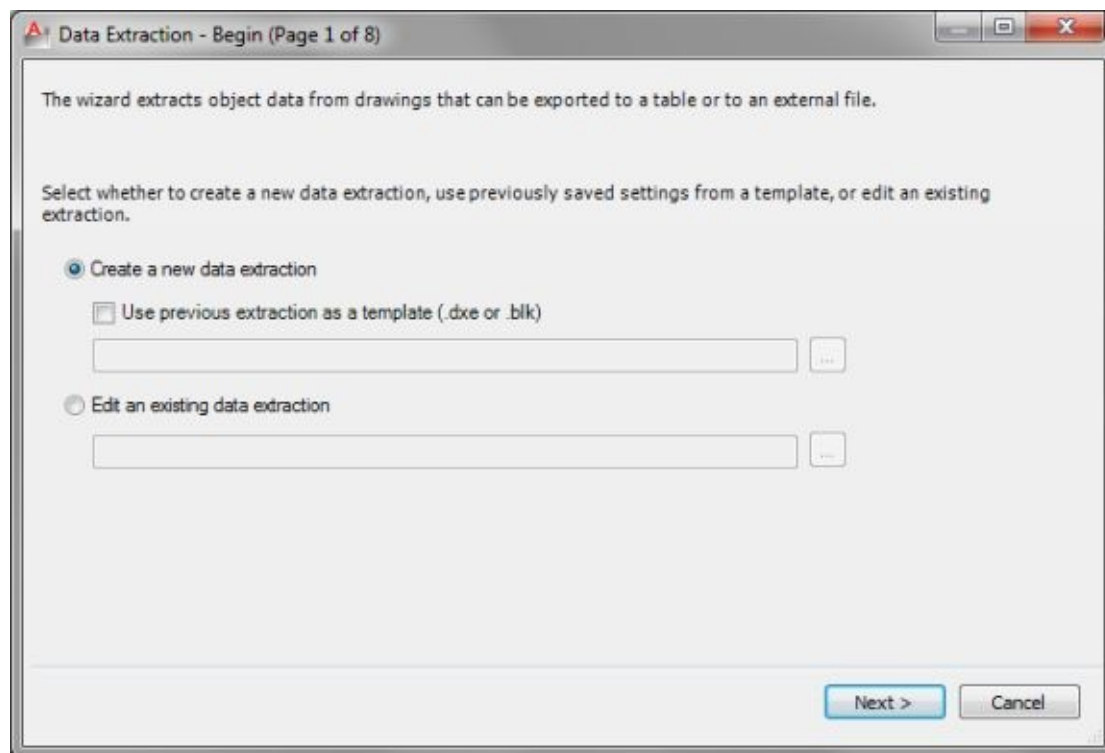


Data extraction is a step-by-step wizard that we can follow easily. We start on the first page.

First page: Begin

In this page, you can start from scratch or select existing data extraction template. If you often create similar column schedule, then you can use it again later.

Because we never do it before, choose to **create a new data extraction** in this page.



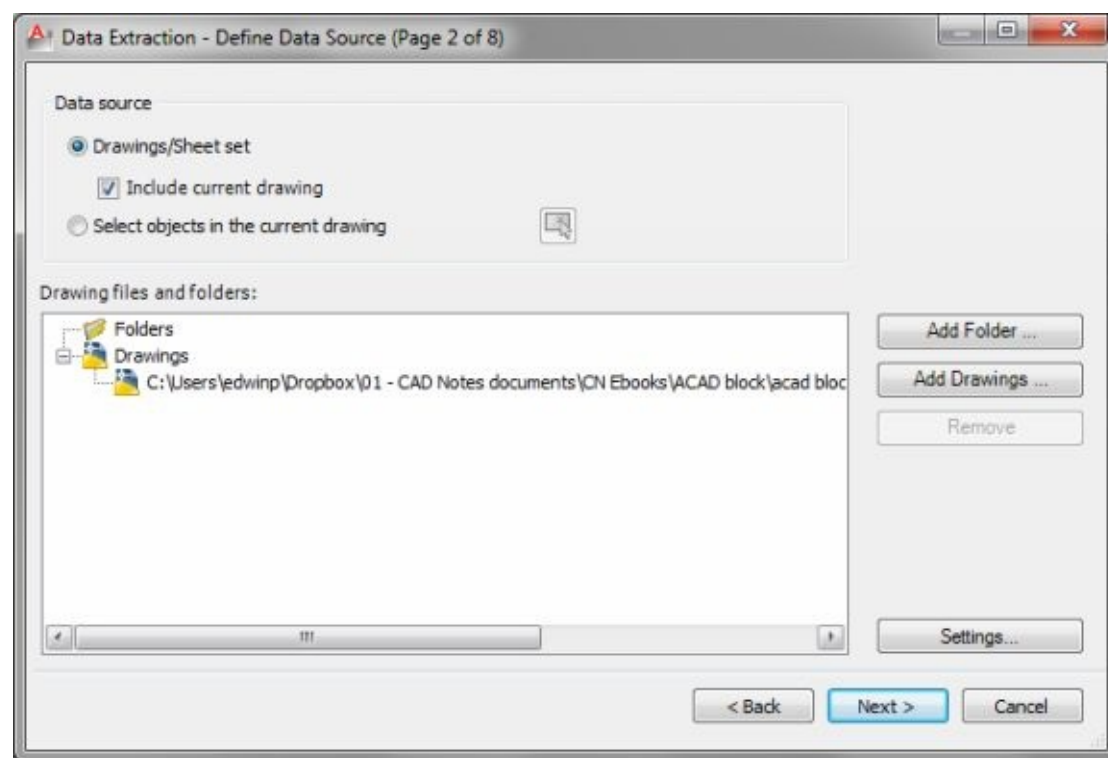
Click next. AutoCAD asks you a file name to save our settings later. Give it name **column schedule**. Click save.

Second page: Define data source

Here, we can define if we want to extract data from all objects in the current drawing file, all objects in

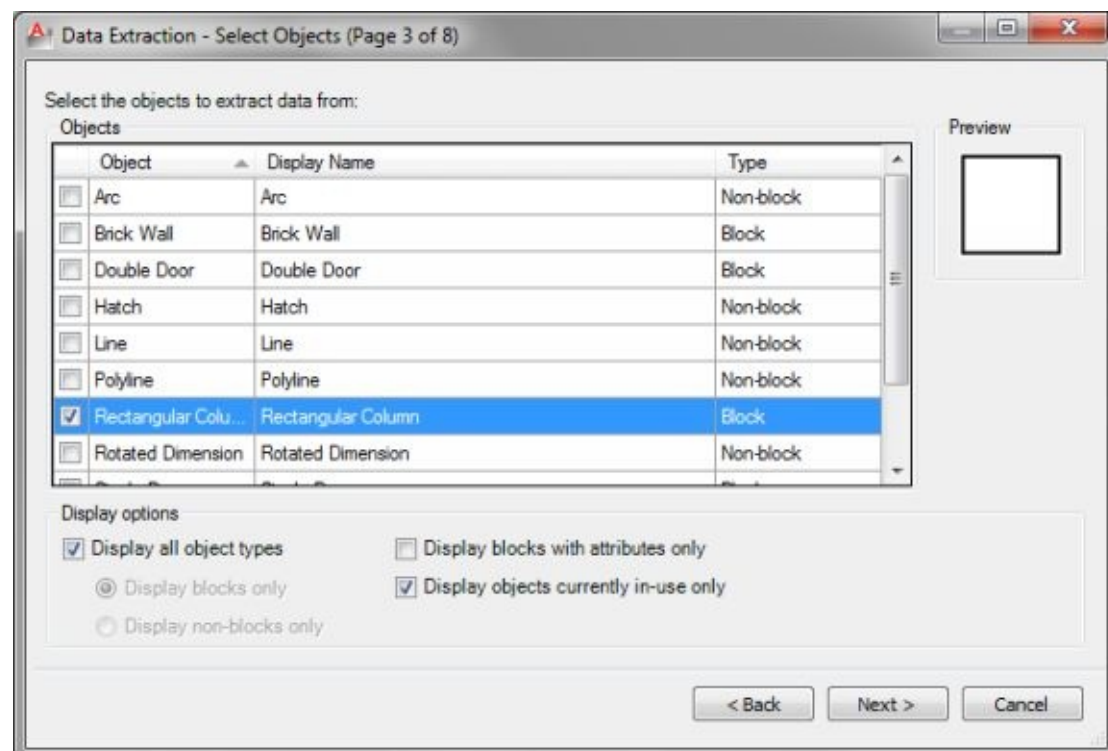
multiple files, or extract data only from selected objects.

For this exercise, choose drawings/sheet set and activate *include current drawing*.



Third page: Select objects

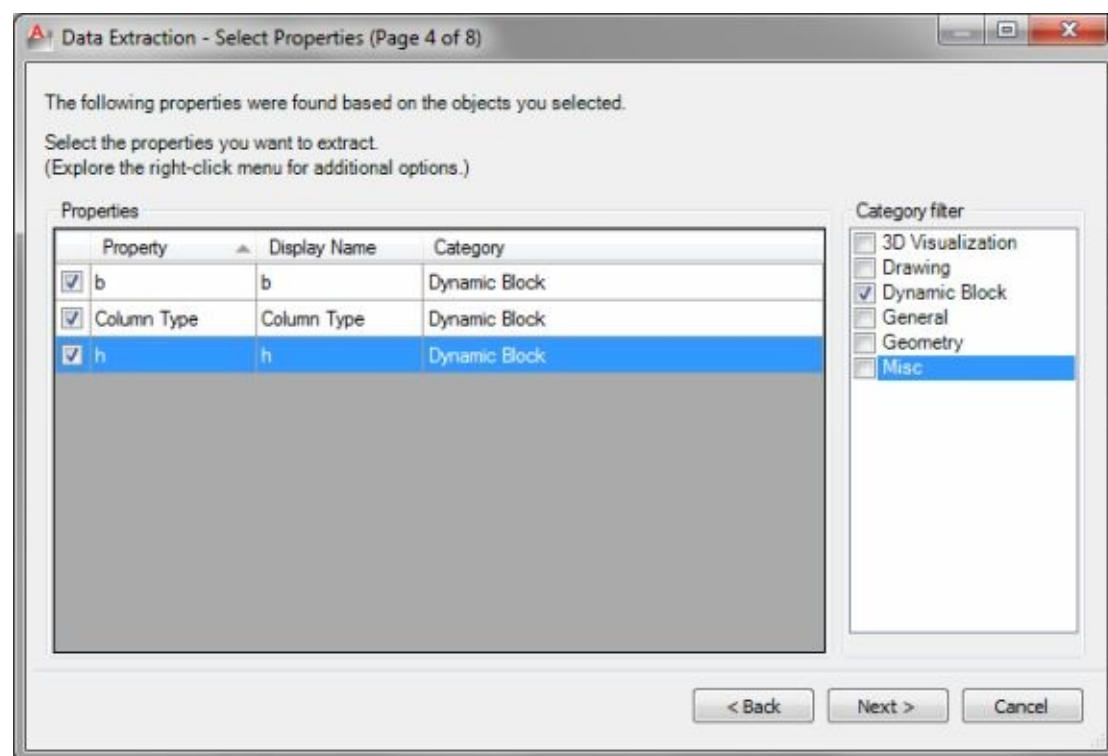
AutoCAD displays ALL objects in your drawing on this page. We only want to create a schedule for the rectangular column. Uncheck everything except rectangular column.



Fourth page: Select properties

The next step is to define which properties we want to extract. We are only interested in getting column type and sizes. They are dynamic block properties. To shorten the list, uncheck everything in category filter (the right column) except for dynamic block.

There are only three properties listed after you filter objects: b, h, and column type. Check all of those properties.

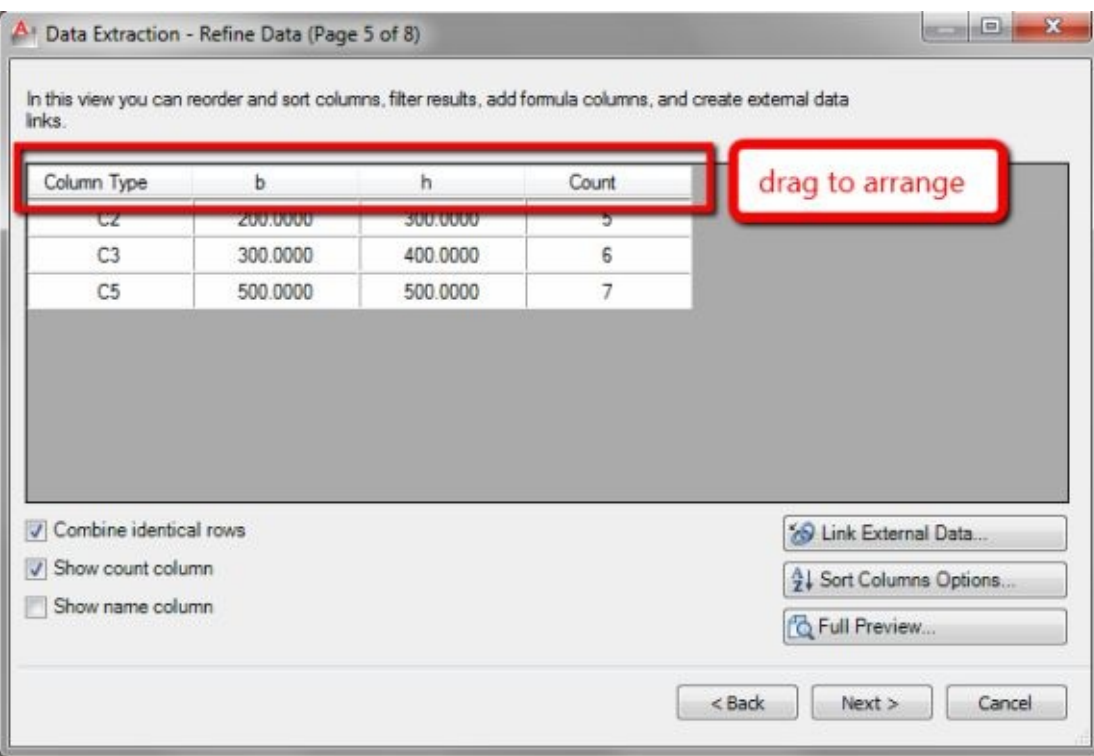


Fifth page: Refine data

In this page, we can see the preview of our table. You can choose to go back and uncheck column you do not need and arrange the column sequence.

Drag column title to arrange them. Uncheck show name column.

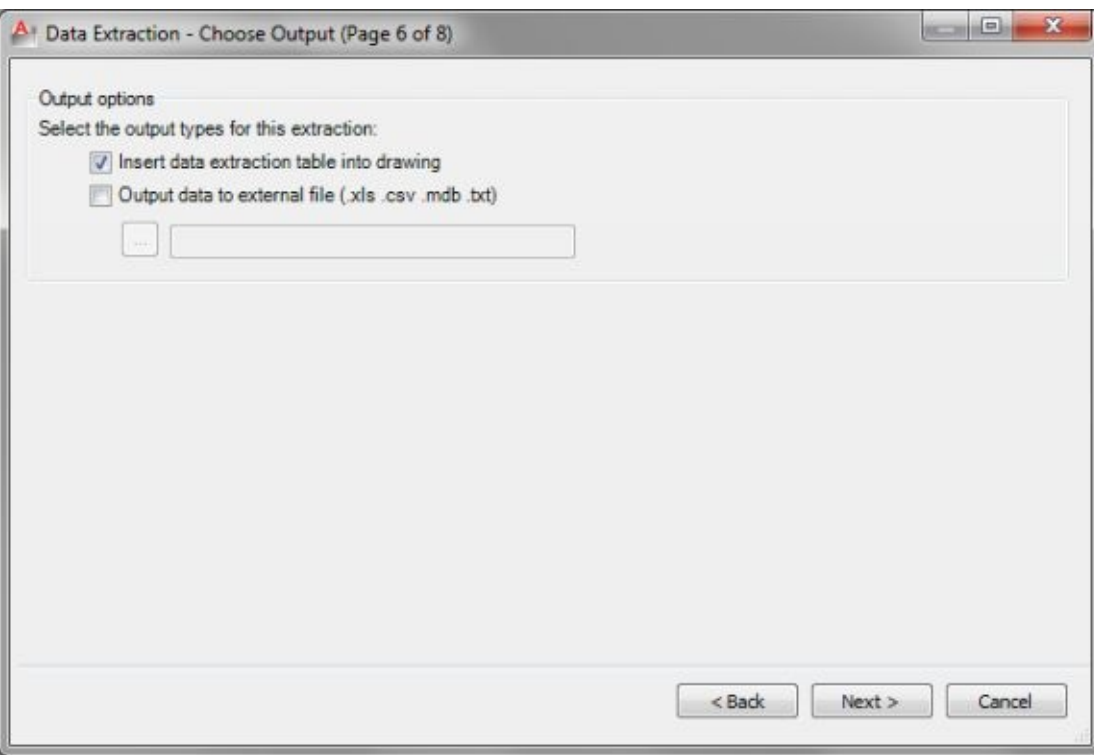
Click next to move to next page.



Page 6: Choose output

In this page, you can choose your output. We want to place this table in our drawing, but you may want to export it to an external file as well.

Click next to continue.



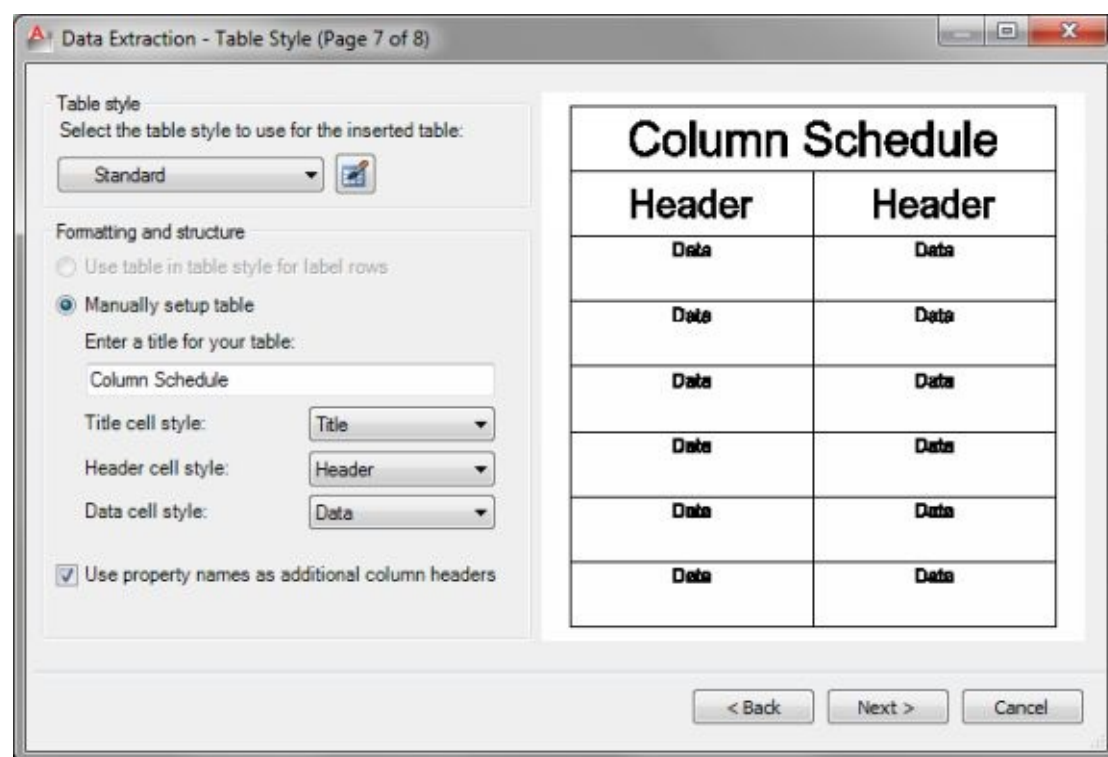
Page 7: Table style

Data extraction output is in a table format. You can choose existing table style or modify it as necessary

here. Standard style is already set for this tutorial.

Click setup table manually and enter your table title. Don't forget to check **use property names as additional column headers**.

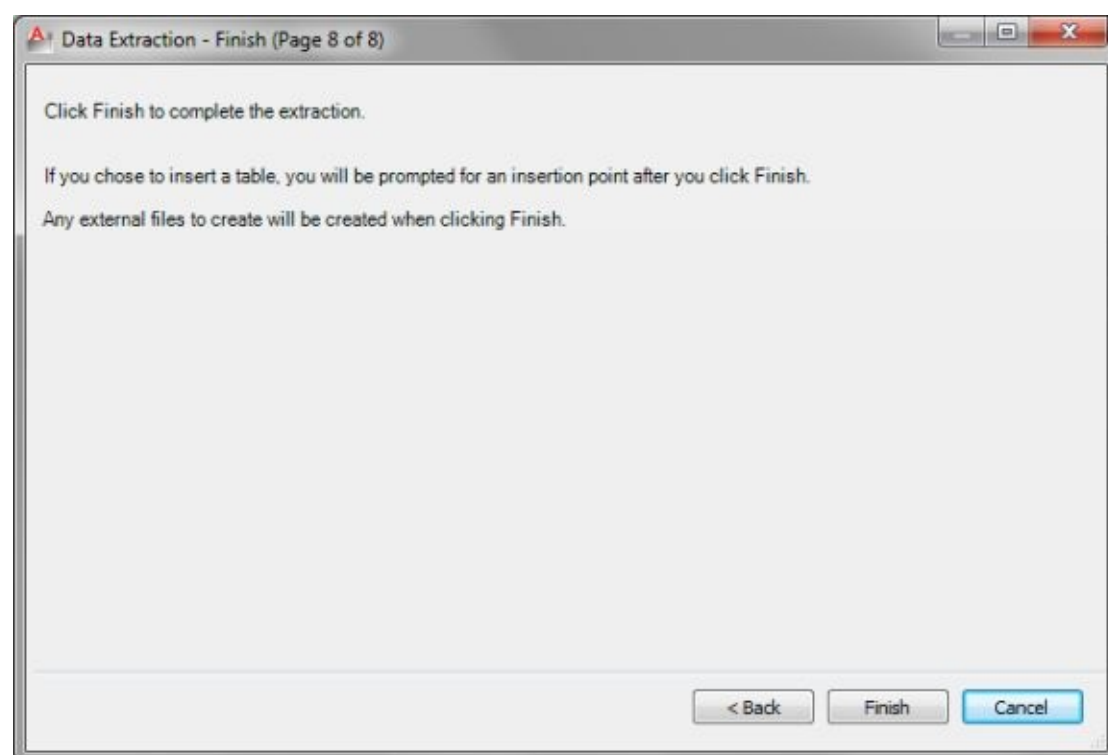
Click next.



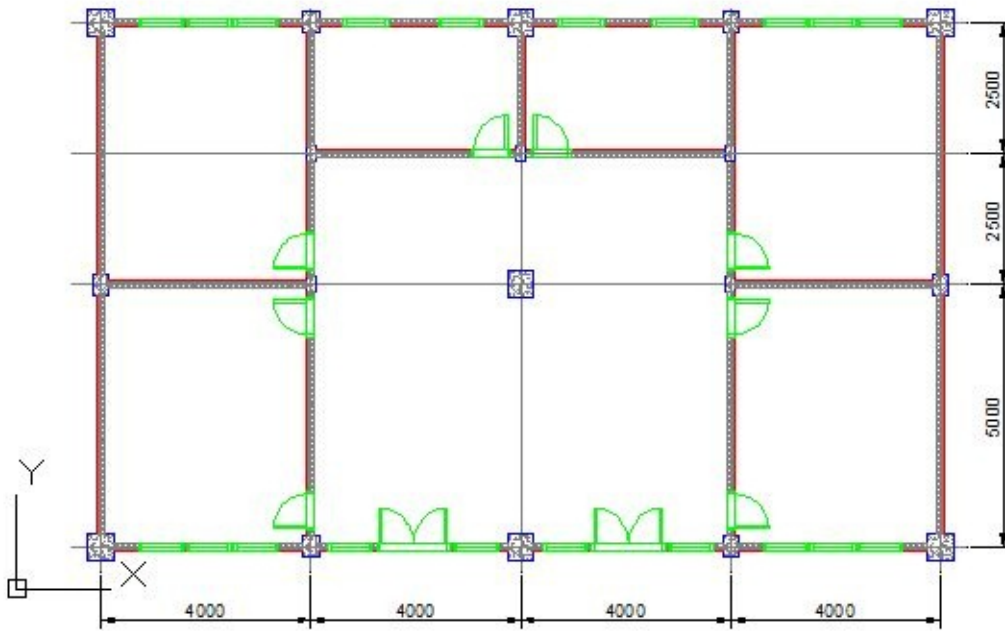
Page 8: Finish

There's nothing to do on page 8. It is just a confirmation page to place or export the data extraction.

Click Finish.



Put your table below the building plan.

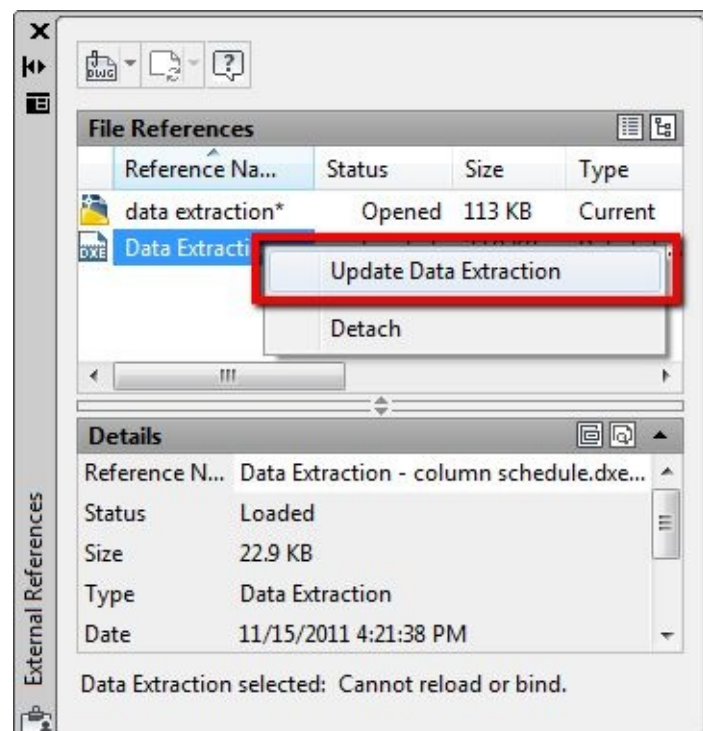


Column Schedule			
Column Type	b	h	Count
C2	200.0000	300.0000	5
C3	300.0000	400.0000	6
C5	500.0000	500.0000	7

For your exercise, try to extract door and windows schedule.

Updating data extraction

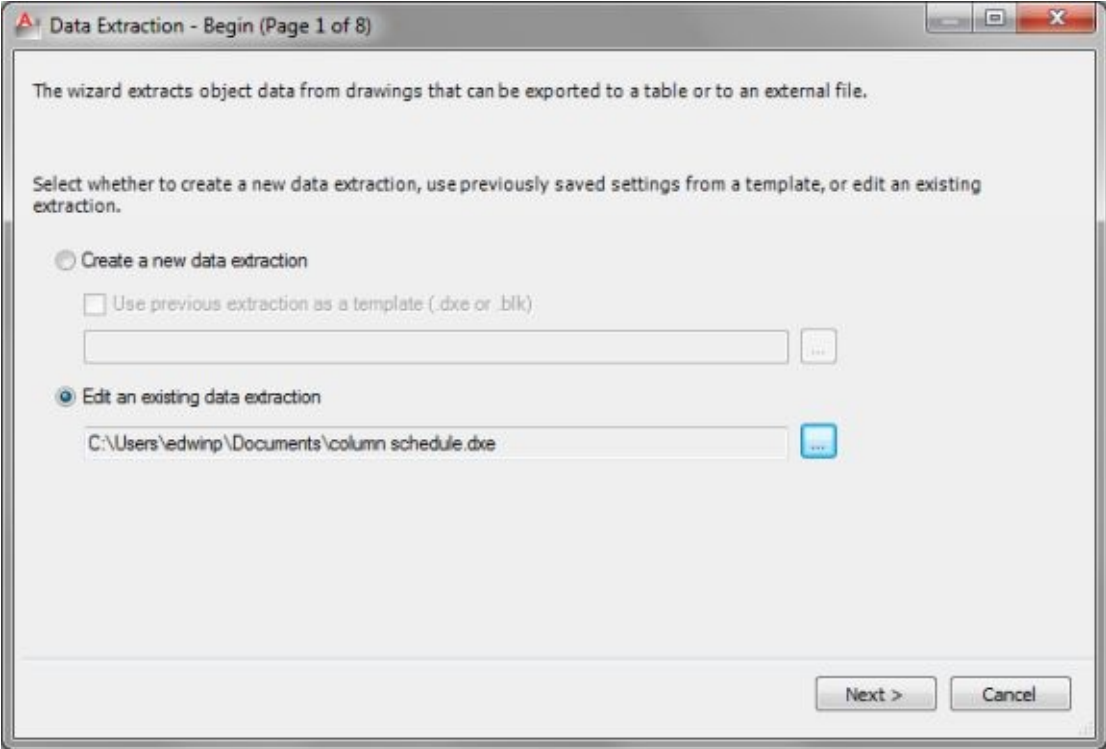
Have you tried to delete or add columns? Your table will not automatically update. Data extraction is recognized as an external reference. To update it, you can open external reference palette, right click the data extraction link and choose update data extraction.



You can detach data extraction to remove the link from your drawing to extraction setup. Your data extraction table is in your drawing, but no longer linked to data extraction.

This method only updates the table in your drawing. If you export it to XLS file, you need to regenerate the excel file.

It is quick and easy. Choose *edit existing data extraction* at the first page; the page 2-7 use our previous setup so we don't need to change them.



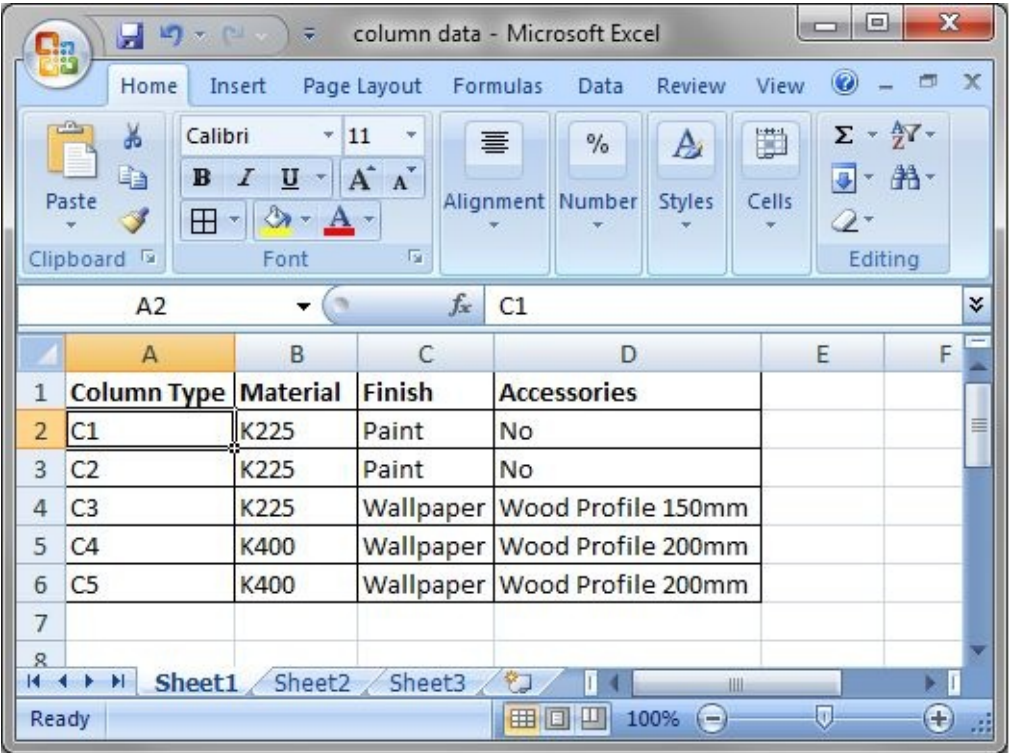
Linking data extraction to external excel file

In some cases, you use the same block with different ID. For example in schematic design. You can it may have many attributes, but they have the same value for identical ID. It does not make sense if you have to define price every time you place a component with the same ID. Remember, prices change regularly. When it changed, you need to update every block attribute in your drawings!

You need to have Microsoft Excel installed on your computer to do this exercise

The internal attribute is very useful if they have a unique value. However, if they have a common value, we better create an external excel file and link it to our data extraction.

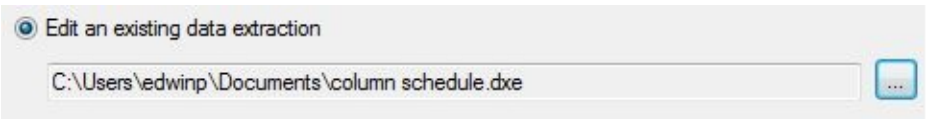
You need to create an excel file that contains the data. In this sample, we still use column schedule. In your exercise files folder, you can find column data.xlsx.



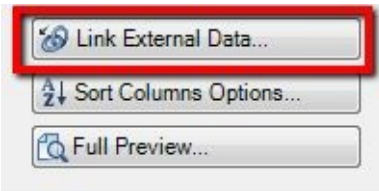
It has *column type* as the identifier and data for each item. The material, Finish and Accessories for each column. We can link it to our data extraction.

First, detach data extraction from external reference palette. Delete our schedule table.

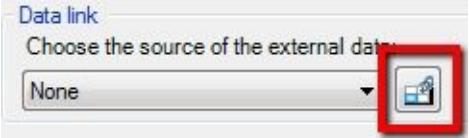
Activate data extraction, and choose *Edit an existing data extraction*. Browse and find our previous data extraction file.



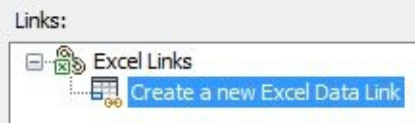
Click next until you find page 5. At the right bottom of this page, click *Link External Data*.



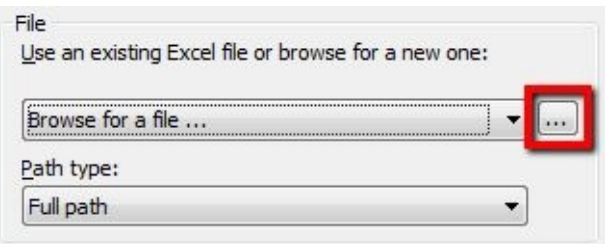
We have not set up a data link, so the list is still empty. Click launch data link manager to create one.



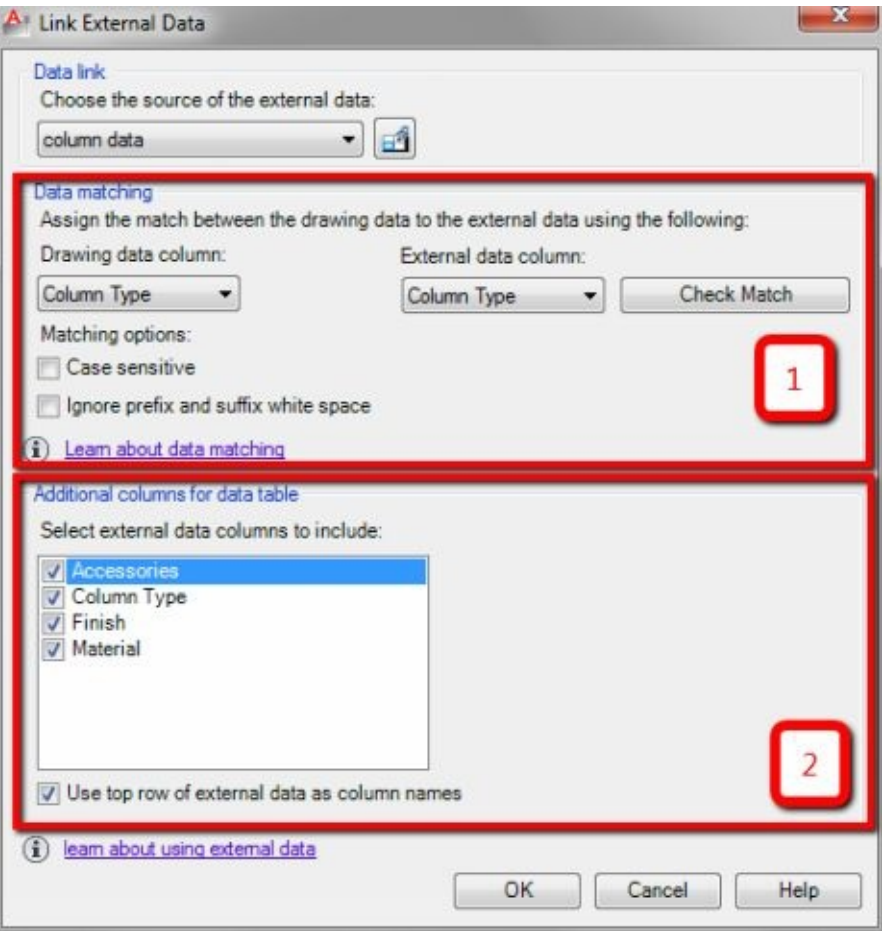
Another dialog box is opened. Click *Create a new Excel Data link*.



Type **column data** when AutoCAD ask you for this data link name, then click OK to accept it. In *new data link* setup dialog, browse and find column data.xlsx in your exercise folder.



Click OK to create the data link. Click OK again to go back to link external data dialog box.



Match data in your drawing and your Excel file (1). Choose column type in drawing data column and also column type in the external data column. You can check if the data in both columns are matched by clicking check match.

You can choose to exclude some columns from the external table here (2). You can see all column in Excel file is listed here.

MORE BLOCK BEST PRACTICES SAMPLES

Until this chapter, you have learned a lot about blocks. Let's explore some samples to give you some more perspective how using blocks can help.

You should already have some ideas how it can help you, but some more examples can give you a better idea of advanced uses of blocks.

Example 1: Building elements sample

In data extraction tutorial, you can find wall blocks and door blocks. If you are interested in learning how to create them, you can find it on CAD notes site:

1. [Creating dynamic block for wall.](#)
2. [Creating dynamic block for door.](#)

Example 2: Automatic Aligning Valve

We created a valve block in the first chapter. When we use it, we often need to rotate it for vertical alignment. After we placed it, we need to trim the pipe line.

We can add some actions to let the block automatically do this.

Alignment parameter

Open valve.dwg that contains your valve block. Open your valve in block editor.

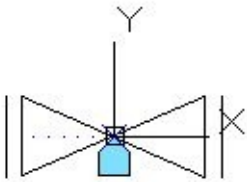
Activate alignment parameter.



Place the alignment parameter at the insertion point. You can type 0,0 to do this or snap to the midpoint of our polyline.

Then type 100,0 to define alignment direction. Alternatively, click a point on the right side of the insertion point.

You have to place alignment parameter at the insertion point. Otherwise, it will not work.



You may want to save this block, close block editor and test it. Try to insert this valve and snap to a vertical line.

Wipeout

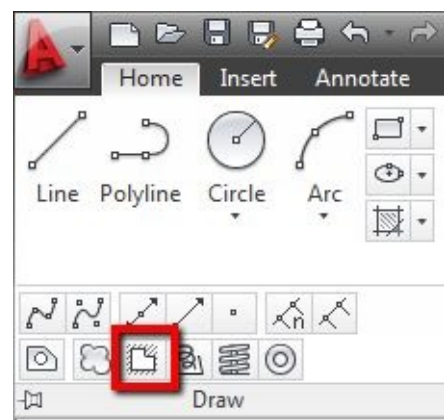
Wipeout is a nice trick to cover objects. It masks object below our valve, so we do not have to trim the pipe lines.

If you have a problem with Wipeout, you can use solid hatch as an alternative. I have seen some problems reported when we plot with wipeout.

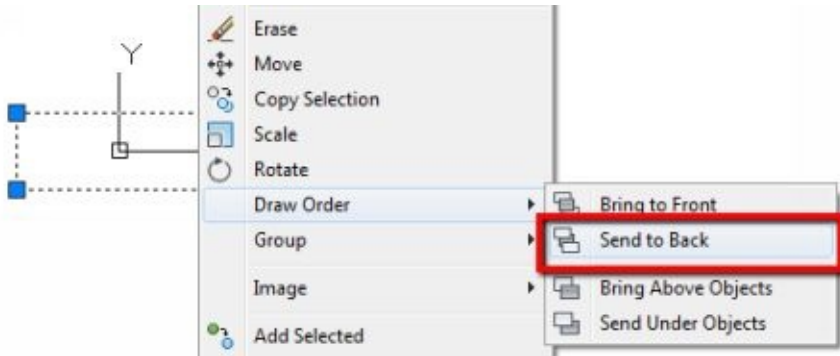
See [this article on CADnotes](#) to see the details.

Open the valve in block editor again.

Activate wipeout from *AutoCAD Ribbon > Home tab > Draw panel*. You may need to expand your panel to see it.



Draw a rectangle using polygon tool from WIPEOUT command. It should cover your entire valve. Select the wipeout, right click. Choose draw order>send to back from the contextual menu.



Now we are done. Save this block and test it again.

Now this block is automatically aligned to vertical lines and 'trim' the lines where you place it.

You can turn off wipeout frame by activating wipeout command and choose frame from the options, instead of picking points.

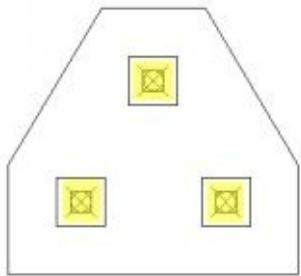
Example 3: Foundation pile cap

In the previous example, we did a little trick so we can draw faster in schematic design. In this case, we add several reference points for reporting purpose. In basic block, we can only report its insertion point. We add more points to report in our data extraction.

Open foundation pile cap.dwg from your exercise files. This drawing is pile cap with three piles. We want to extract the pile coordinates for setting up.

Add reference points

Add three points in the middle of our rectangle. These points are our attributes reference.



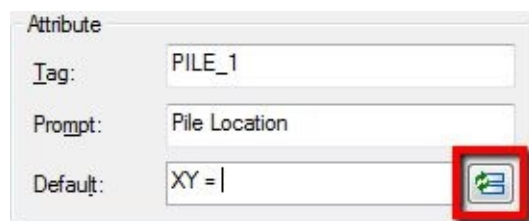
Hint: You can see points clearly by changing point style. Type DDPTYPE then [enter].

Adding attributes

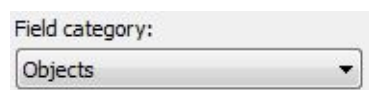
Now add an attribute.

In attribute definition dialog box, check preset. We want AutoCAD to set this attribute value automatically, and we do not want users to change it. AutoCAD does not ask users to input this value if we activate preset. Also check invisible, because we do not want to show this attributes. This attribute is only for reporting purpose.

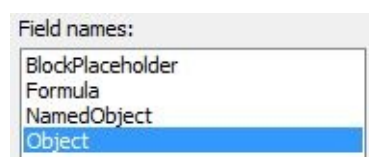
Fill attributes fields as necessary. For default value, type XY = then click insert field.



In opened dialog box, change field category to objects. We want to get properties from a point.



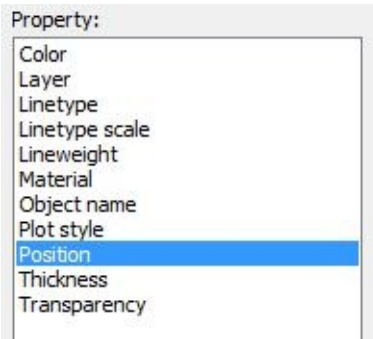
After you changed the field category, AutoCAD filters fields and show only objects fields. Choose *Object* from the list.



On the middle column, you need to choose which object to get the property. Click Select Object.

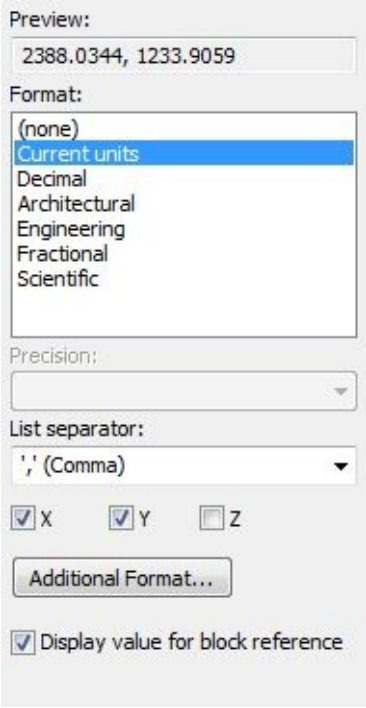


Pick the top most point. After you pick the point, AutoCAD updates the property list. Choose position. This property is the point coordinate.



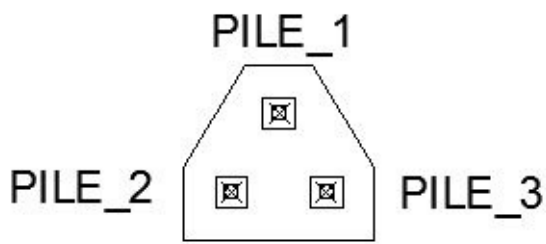
On the right column, now you should see how the properties are displayed. You can change the format and precision here.

Below, you can choose which value to show. Uncheck Z value. We only use this drawing in 2D, so we do not need the Z value.



Click OK to use this field. Click OK again to create the attribute. Place the attribute at the top of our pile cap.

Repeat the process for pile no.2 and 3.



Add one more attribute. Create a regular attribute as pile cap ID.

Create them as a block. You can use 0,0 as the block base point.

Now the block is ready. You can place the block to your foundation plan. You may create your own as exercise. Alternatively, you can use the one provided in final folder. Open foundation pile cap – final.dwg. This file contains the same block in foundation plan.

Try to use data extraction to report all foundation coordinate. You can use my data extraction settings (included in exercise files) or try by your own.

Data Extraction - Refine Data (Page 5 of 8)

In this view you can reorder and sort columns, filter results, add formula columns, and create external data links.

PILE_ID	PILE_1	PILE_2	PILE_3
01	4605, 1269	4305, 769	4905, 769
02	9105, 1269	8805, 769	9405, 769
03	13605, 1269	13305, 769	13905, 769
04	4605, 7269	4305, 6769	4905, 6769
05	9105, 7269	8805, 6769	9405, 6769
06	13605, 7269	13305, 6769	13905, 6769
07	4605, 13269	4305, 12769	4905, 12769
08	13605, 13269	13305, 12769	13905, 12769

Combine identical rows
 Show count column
 Show name column

[Link External Data...](#)
[Sort Columns Options...](#)
[Full Preview...](#)

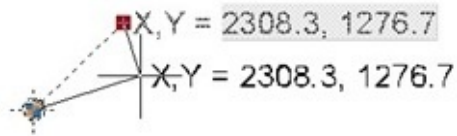
[< Back](#) [Next >](#) [Cancel](#)

Example 4: Coordinate label

This example is one of the most popular block tips in [CADnotes](#). This tip is similar to pile cap foundation. Only here, we show the attribute to label the coordinate.

To add some style, we add a leader and add stretch action to allow users to move label position.

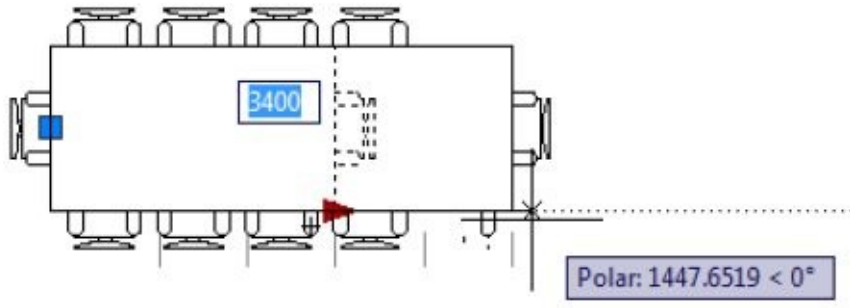
Create this block by your own. Use fields, so when you move the block, coordinate shown is updated automatically.



If you have a problem with it, you can [refer to this post on CADnotes](#).

Example 5: Adjustable table length

This example is another exercise for you. This table also uses stretch action. We also add array action for the chair. So when users lengthen the table, the chairs are automatically added.



Again, try to create it by yourself. You can get the chair block from AutoCAD sample files. You can find the finished block in sample folder.

MANAGING AND SHARING BLOCKS

You created great blocks. Now we need to manage them so our team and we can use it.

Managing block files

We need to save blocks in separate files. We can add all block definitions in a template, so when we create new files, they are already set. However, this is not recommended because it makes your files full of blocks that you do not use. It can make your files large and prone to corrupt. It is better only to insert blocks that you need.

Single block in single file

This method is common in the past, and many people still use this method. You have learned that you can draw in a file, then when you insert the file as a block, AutoCAD converts it into a block. To manage similar blocks, you can save them in a folder.

This method is preferred because it is easy to insert that file as a block when you create a simple AutoLISP program.

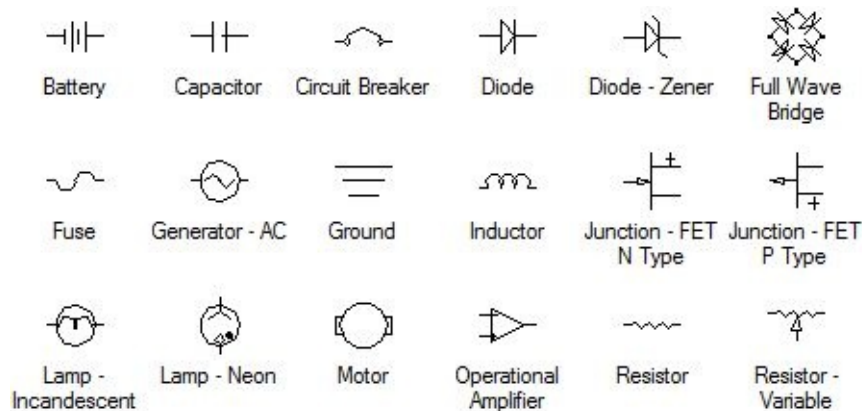
These are some examples how you can insert a file as a block using AutoLISP:

1. [Block and conditional if: elevation label.](#)
2. [System variable and conditional COND: grid label.](#)

Multiple blocks in single file

The better way to manage your blocks is to save all similar blocks in a single file. In AutoCAD, open the Design Center. You can see some sample files here.

They are *block libraries*. Each file has several blocks in the same category.



This method makes fewer files to manage. Using design center makes it very similar to managing a block in a single file. You can see the block and insert it to your drawings.

You can also easily create tool palettes from each file and use it to insert your blocks.

Creating and using tool palettes

What is a tool palette? Snipped from the help file:

Tool palettes are used to manage blocks, hatches, and other custom tools

Block is not the only object it can handle. However, we only focus on blocks.

Adding blocks to tool palette

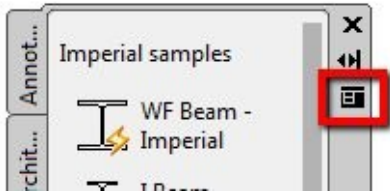
Palette is very useful and extremely easy to use. We can add an object to palette by drag it and drop it to the palette. You can create several palettes to separate blocks from different categories.

Let's create a new palette for this exercise.

Creating new palette

Open tool palette. Press [ctrl] + 2 if it hasn't opened. Right click anywhere in your palette. Choose new palette from the context menu.

If you have a problem using contextual menu, you can click the Properties button and choose new palette from the menu.



Give your palette name.

Open sample files or any file that has blocks in it. You can use a sample from design center or exercise from this e-book.

Click a block in your drawing once. This action selects the block. Now click selected block again and hold your mouse button. Drag it to your palette. You can see the block in your palette now. Very simple, isn't it?

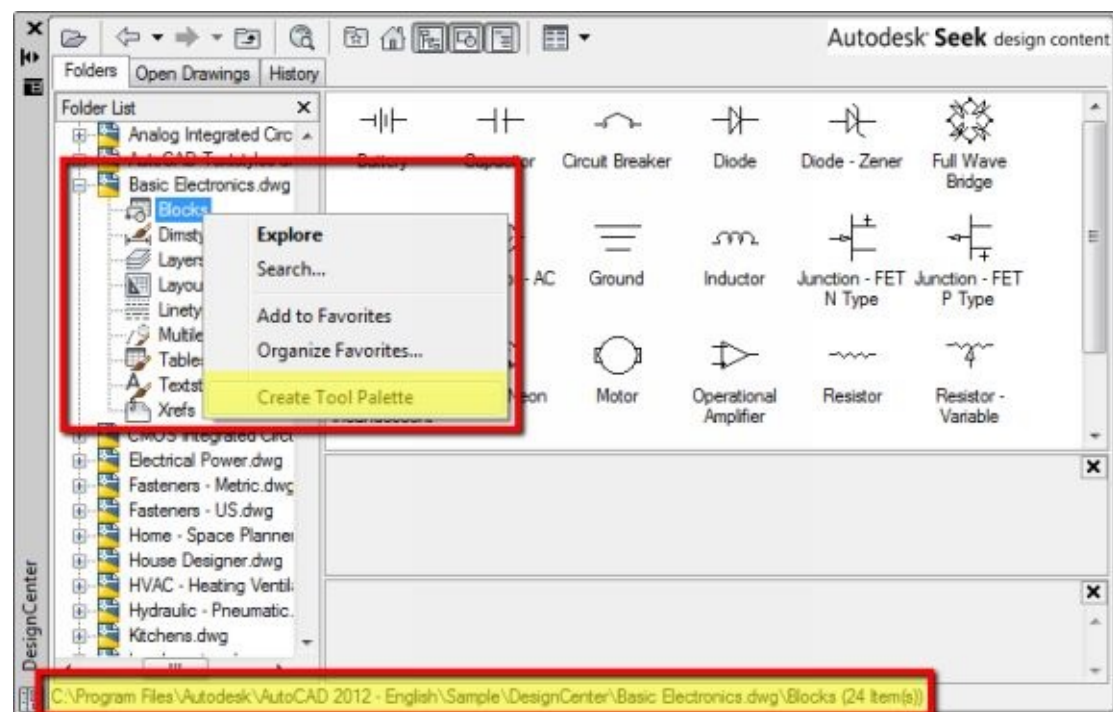
Palette does not save your block definition. It only provides a link to the original file and a thumbnail. Do not delete your file that contains the block. If you do, you lose your blocks!

It is better to save it to a shared folder on your network.

Creating palette using Design Center

The previous method is easy. However, if you have hundreds of block, it can be a tedious task. The next method is another benefit of placing multiple blocks in a single file.

Open Design Center. Navigate to design center sample files. Refer to the folder location in design center dialog below.



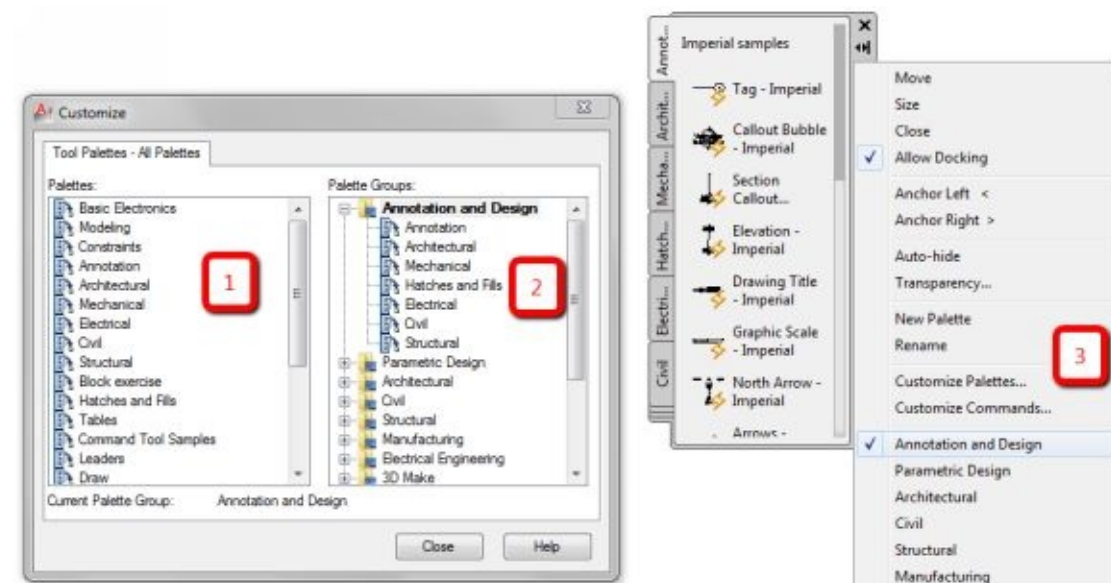
Click the (+) sign to expand a file. Click block category to see the blocks. Right click on file name (or block category), and choose to create tool palette. AutoCAD creates a new palette, give it the same name as the file name. AutoCAD put all block definitions in that file to the new palette.

Creating palette group

We can use the palette to manage blocks in same categories. However, when we have dozens of palettes, it can be difficult to switch between palettes.

When we are doing a specific task, we can show only palettes we need, and hide the others. For example, when you are doing a civil drawing. You do not use architecture tools. Only civil drawing objects, annotations, and other that relate to your task. This reason is why we use Palette Group.

Right click your mouse above tool palettes and choose *Customize Palette* from the contextual menu. Alternatively, you can access it through properties button.



There are already several palette groups available (2). See below annotation and design group. There are several palettes available. Activate the group.

You can do it by right click the group in Customize Palette dialog, and choose set current. Alternatively,

you can click Properties button above tool palette title, and choose from the list (3).

You can see only palettes in that particular group. Try to change current palette group to another group. Examine what happened to your tool palette.

You can add a palette to a group by dragging it from left column (list of all available palette) to below a group folder. To remove it, you can drag a palette from the right column to the left. Alternatively, you can right-click a palette and choose remove from the contextual menu.

Sharing your blocks to your team

After you create your blocks, you need to consider how you should share it with your team. You can use your blocks as standard libraries. It is not a good practice if anyone in your company can create their blocks.

The best place to share your blocks is on your company server. You might need to ask IT department to create a shared folder for your team. You need to consider different permission for your users. CAD managers should have administrative access because they need to update the blocks regularly. Your users only need read-only permission.

You can create and share tool palettes too. However, remember, tool palettes only point to the block location. So every user needs to be able to access the same path.

The easiest way to do it is to allow users to create their palette. They only need to find the block library on the server using design center. Then they can create palettes by a few clicks.

There are many ways to share your blocks. You can use AutoLISP or .NET programming if you are good at it. Find one that you think works best for you, and try not to overdo it and waste your time.

AFTER YOUR FINISH THIS BOOK

Blocks can significantly increase your productivity. Try to understand how it can work and how It can help in your daily tasks.

You may not use all of the features. Moreover, the samples in this e-book may not reflects your need in your industry. However, you should understand how blocks may help you, and implement it in your company.

If you have questions or want to discuss more blocks, you are welcome to discuss it on our support forum. Here are how you can contact us:

1. Our Facebook page: <http://www.facebook.com/CadNotes>
2. Our Twitter page: <http://twitter.com/CADnotes>
3. Our Google+ page: <http://google.com/+Cadnotestips>