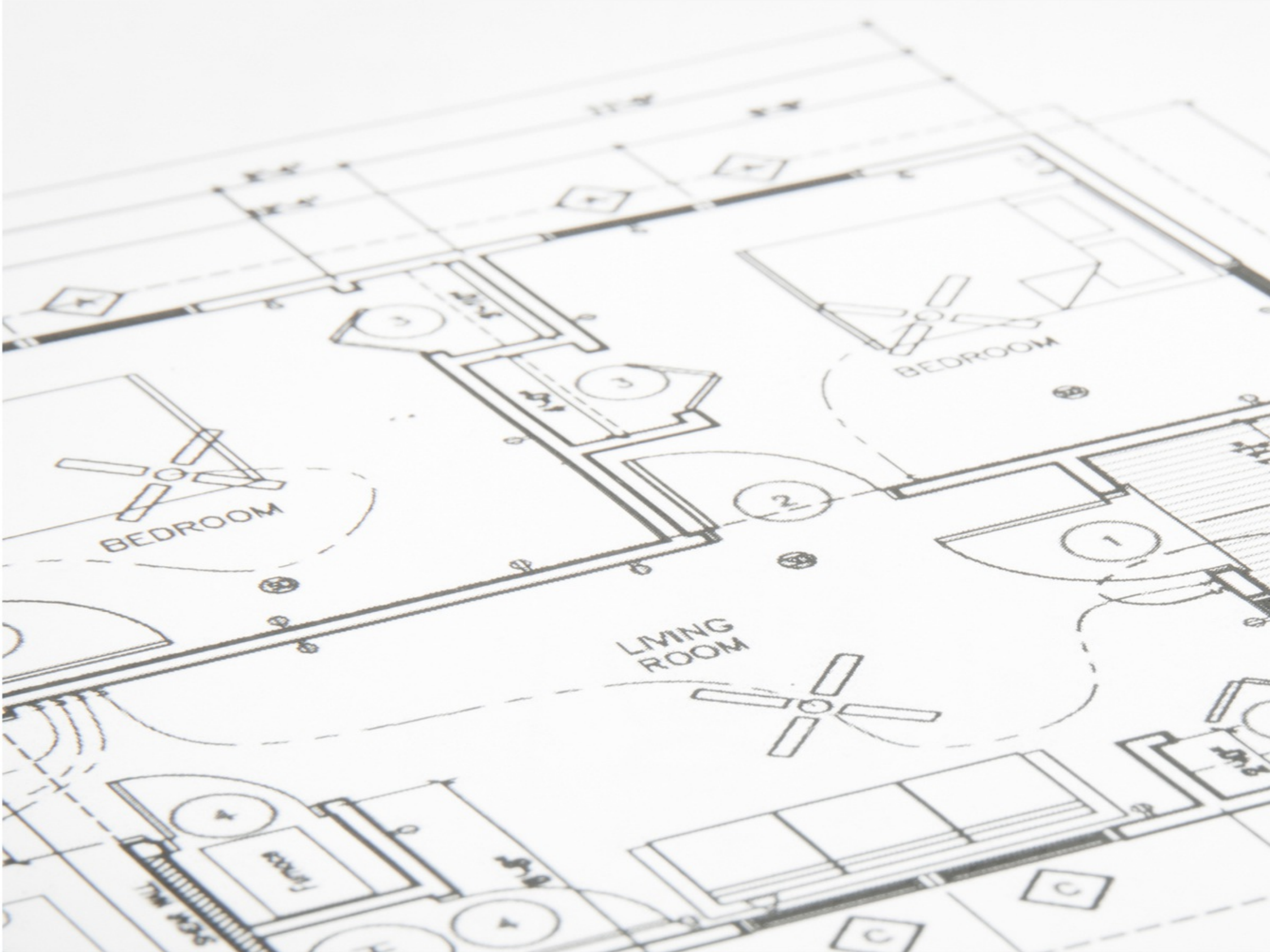


THE ART OF USING AUTOCAD 2D

FOR MECHANICAL DESIGN ENGINEERING

PROFESSIONAL 2D DRAWING DESIGN METHOD



CADMAGIC

The Art Of Using AutoCAD 2D For Mechanical Design Engineering

Professional 2D Drawing Design Method

-- By CADMagic --

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Introduction

Model Space is a great tool of AutoCAD, specialized for presentation and printing of drawings. This particular book not only teaches you how to work with Model Space, but also gives you an overview of the standard steps for developing all AutoCAD Mechanical drawings.

With this book, you will completely master Model Space, manage drawings scientifically, and represent drawings with different scales also become extremely easy and fast. I will guide you in detail on how to combine tools in AutoCAD which are Dimstyle, Textstyle, MleaderStyle, Annotative, Annotation Scale, and Ltscale for you to apply in file management and batch printing.

The core value that I convey to you through this wonderful book is the word “Process”, having everything in your hands. To work effectively with AutoCAD and Model Space, you must have Processed.

I am sure that, after you practice with this book, from now on you will give up your old habits and switch to the Annotative, Annotation Scale because of the amazing results you see it brings. Please practice with my book. The value of a book is your productivity!

This book does not go into detail teaching you how to build a view, nor does it show you how to use commands. There are already too many technical books in-depth about the rules of drawing representation or the rules of projection construction. Moreover, today, projections can no longer be constructed on AutoCAD but switch to 2D export directly from 3D drawing software. But "exporting the projections " to AutoCAD and developing 2D on AutoCAD is still very popular.

Your task is to export 2D projections to AutoCAD and annotate dimensions, note technical requirements, present and manage them on AutoCAD in a scientific way. Then this book will help you with that! Because I have not seen any book on the market written on that topic yet, while it is very practical and necessary. A person who has worked for a long time, even for more than a decade, is unsure how to manage it to be streamlined and convenient for printing later.

Chapter 1: Technical Drawings and Projection Methods

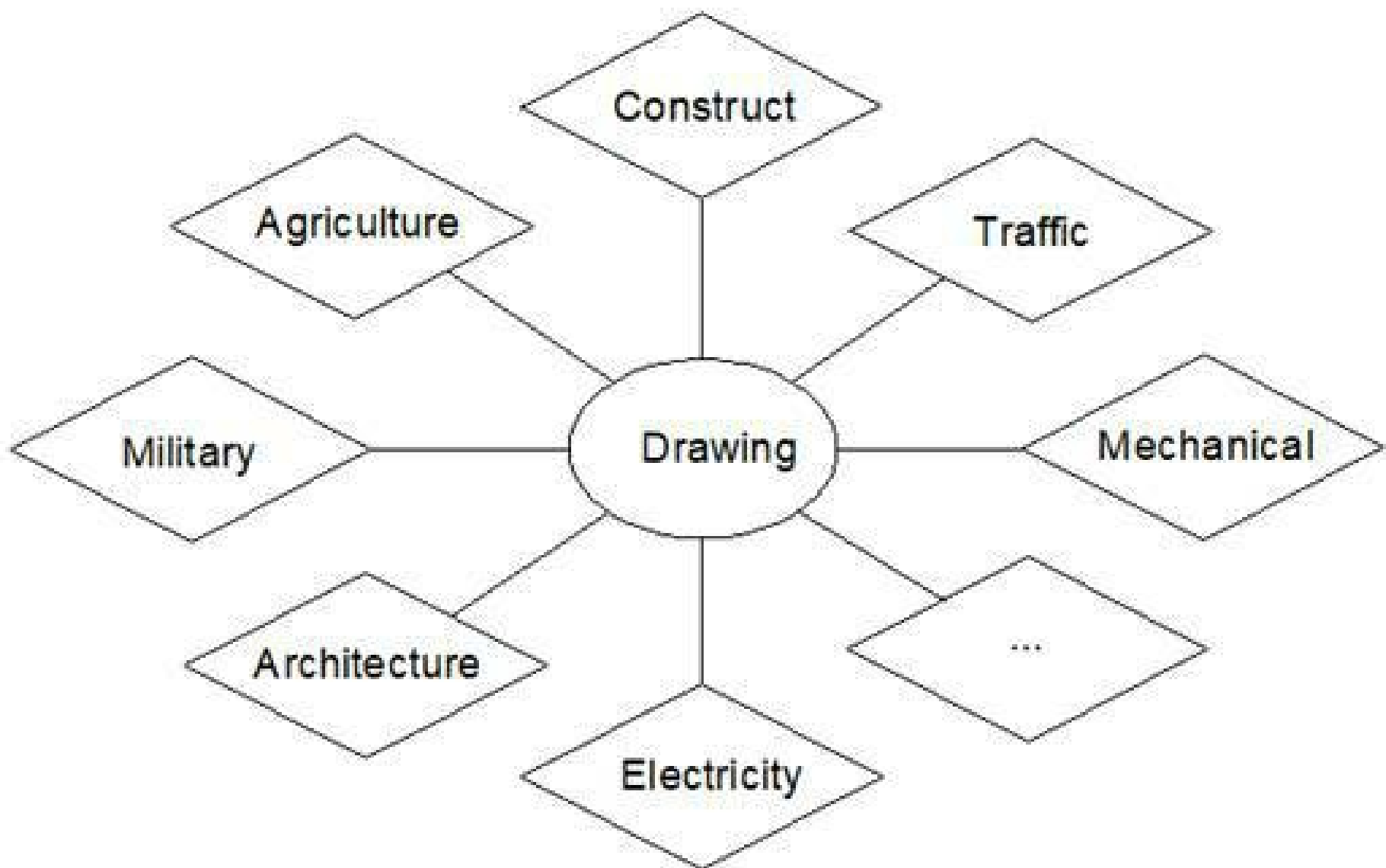
1. Technical drawings

When you think of drawings, you probably picture in your mind an image of the projections placed neatly on a white background, with a name frame along with numbers, size notes, and some notes. text.

Understandably, drawing is a language used to communicate and convey information between technical people.

Drawings of objects are presented as 2D views in the projection directions according to a general rule. From there, the drawing reader can visualize the shape, structure, and size of the object.

Classification of drawings:



There are two most important types of drawings in engineering:

- Mechanical drawings: related to the design, fabrication of assembly, use of machines and equipment
- Construction drawings: related to the design, construction, and use of architectural and construction works.

2. What is the projection method?

Drawings are technical languages, and projection methods are ruled that express that language. The projection method specifies the order in which the object's 2D projections are arranged on the drawing.

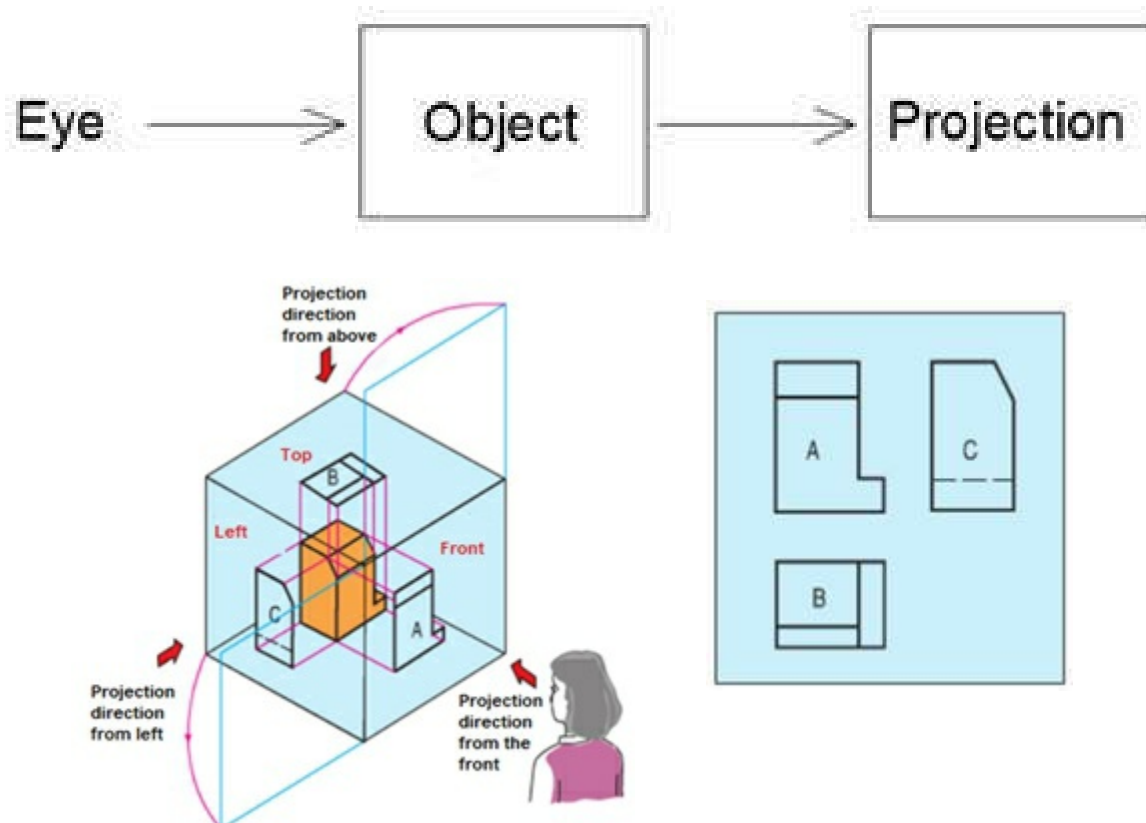
Currently, there are two methods of projection in the world:

2.1. First angle projection method (ISO standard)

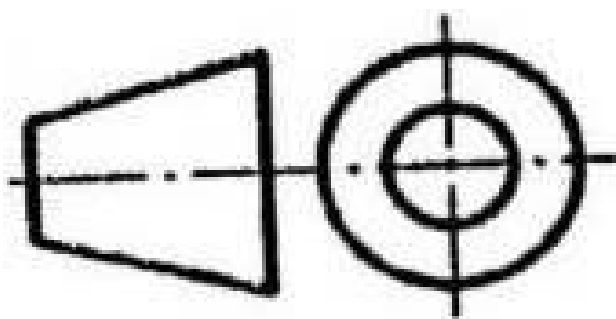
The first angle projection method is based on ISO standards.

In the first angle projection method, the object placed between the observer of the coordinate plane is the projection plane on which the object is projected perpendicularly onto it. The first angle projection method is based on ISO standards.

The first angle projection method characteristic: where the human eye is located, the projection is located on the other side of the object.



Characteristic symbols of the first angle projection method:

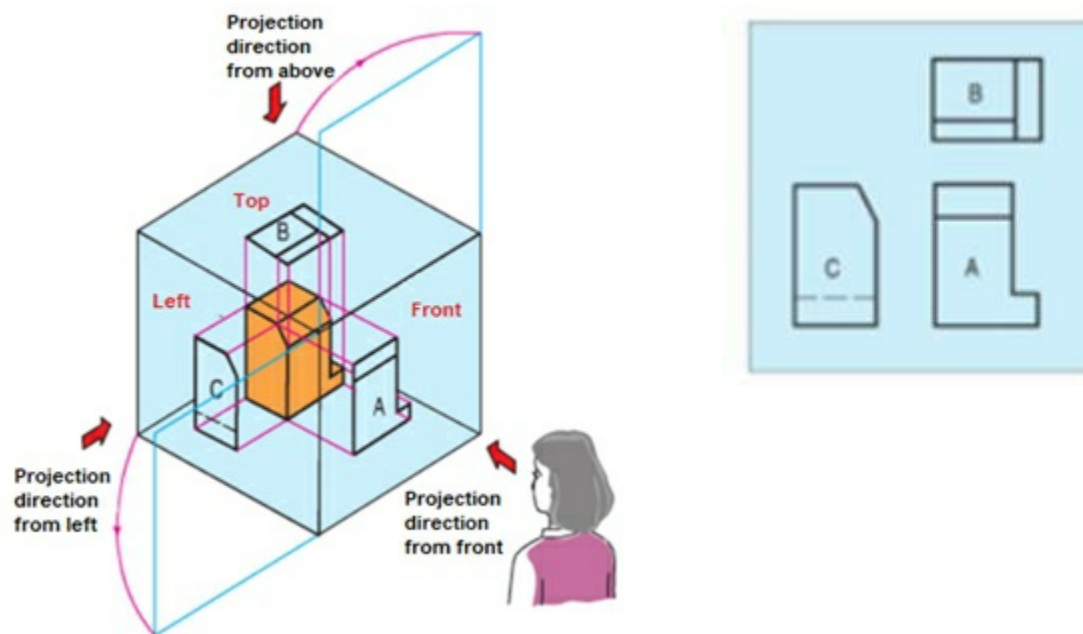


2.2. Third angle projection method (ANSI standard)

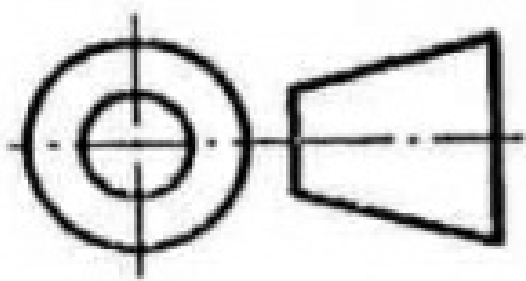
The third angle projection method is based on the U.S. ANSI standard (the Japanese JIS standard also uses a third-angle projection method)

In the third angle projection method: coordinated planes are placed between the observer and the object, the object is perpendicularly projected onto those flat tiles. On each plane, the projection of the object is the same as the one that the observer sees when looking directly at the remote angle into the transparent projection plane.

Third-angle projection method characteristics: projections are placed between the human eye and the object.



Characteristic symbols of the third projection method:



3. Technical drawings in the mechanical industry

For mechanical AutoCAD, there are usually types of drawings including:

- Part drawing: a single detailed drawing that fully shows all the characteristics of an object.
- Assembly drawing: drawing of assemblies assembled and meeting the technical requirements and characteristics of the assembly, together with a list of details participating in the assembly.
- Explosive drawing: drawing of a detailed assembly in a disassembled state, ready to be assembled, together with a list of details participating in the assembly.

Normally, to do a complete set of design documents, you will have to go through all of the above drawings.

Chapter 2: Numbers Can Lie

Types of Ratios on a Technical Drawing

Have you ever wondered, "*How many types of ratios are there on the drawing?*"

Here's the answer!

On a drawing, the following scale types exist:

- Real draw scale
- Drawing scale (scale of the drawing)
- Detail section scale (Real draw scale of the Detail section)

I see a lot of people don't understand these concepts. That's why I had to put it at the beginning of the book. A very important background content!

1. Real draw scale

Have you ever heard anyone talk about the **Real draw scale**?

Let's be honest!

Most likely not yet!

Because not many people notice and understand about it. Technical books are updated too slowly compared to current software engineering. But it does exist, it exists side-by-side with AutoCAD software. You can consider this as a new concept also.

Real draw scale is only meaningful in Model space, it has no meaning after being printed on paper. People often ignore the Real draw scale because they are being drawn correctly, ie always drawing on the AutoCAD Model with the true size of 1:1. The rule of representing objects in real size (Real draw scale = 1:1) is also a mandatory rule when representing drawings in Model space.

For example, a line segment in real life has a size of 100mm, on Model space you draw a segment that has the "true size" of exactly 100mm and you write that size (display size) out as 100mm. At that time, the Real draw scale is 1:1.

Don't believe numbers if you're just "seeing"!

Be careful my friend!

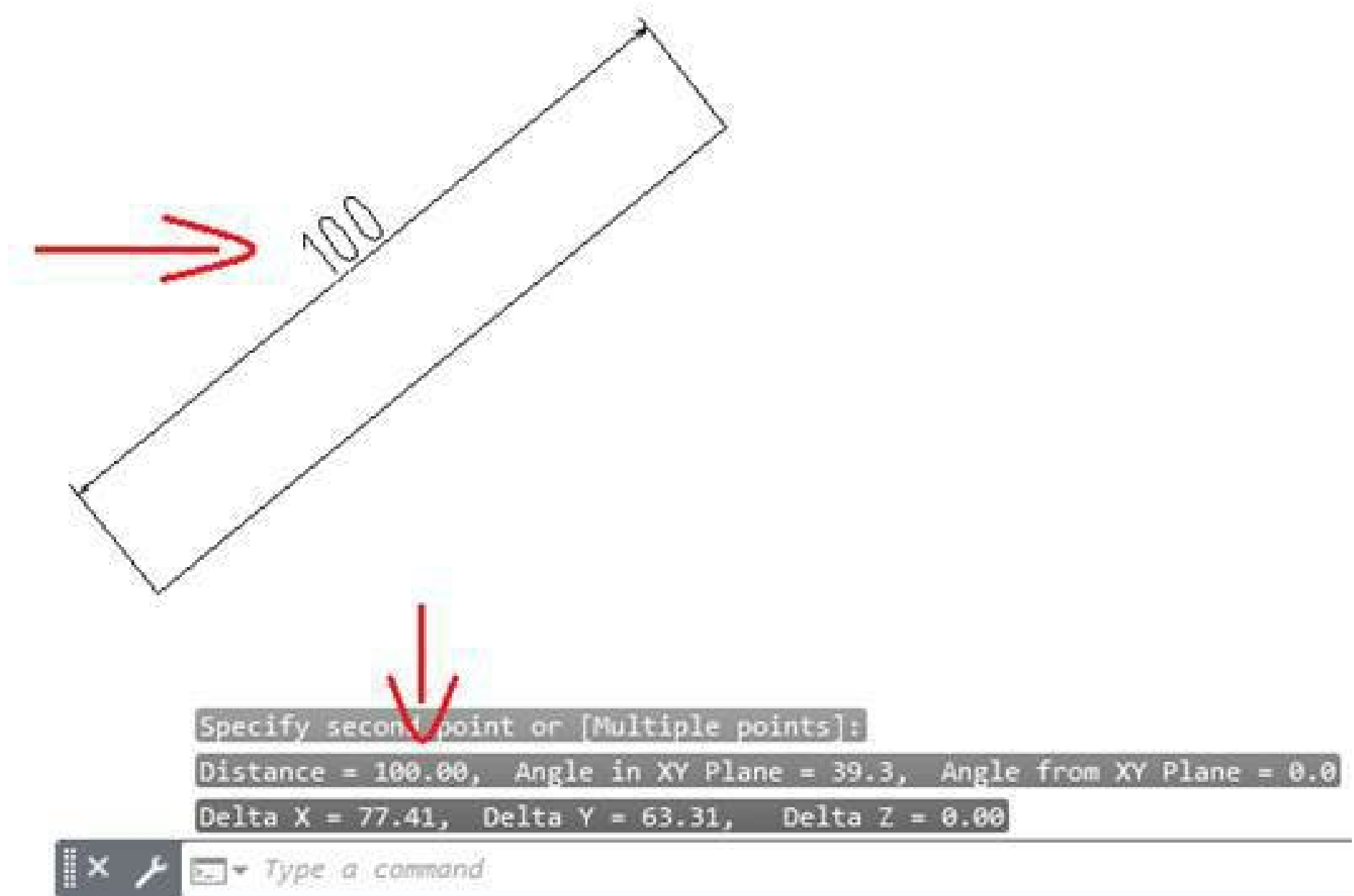
A real line segment is 100mm long, on Model space is note the display dimension (D-Display) as 100mm, but if its true length (R-Reality) on Model space is not 100mm. At that time, they were drawing with a Real draw scale **that was not 1:1**

- To check if the last segment is exactly equal to 100mm or not?
- You type the command: **DI** ↙ (DIST)
- Click on the 2 ends of the line segment
- Check how much the **Distance** value appears on the screen.

If Distance = 100: they are drawing at true size (with Real draw scale = 1:1).

Case 1: The drawing person is drawing correctly, with the Real draw scale = 1:1

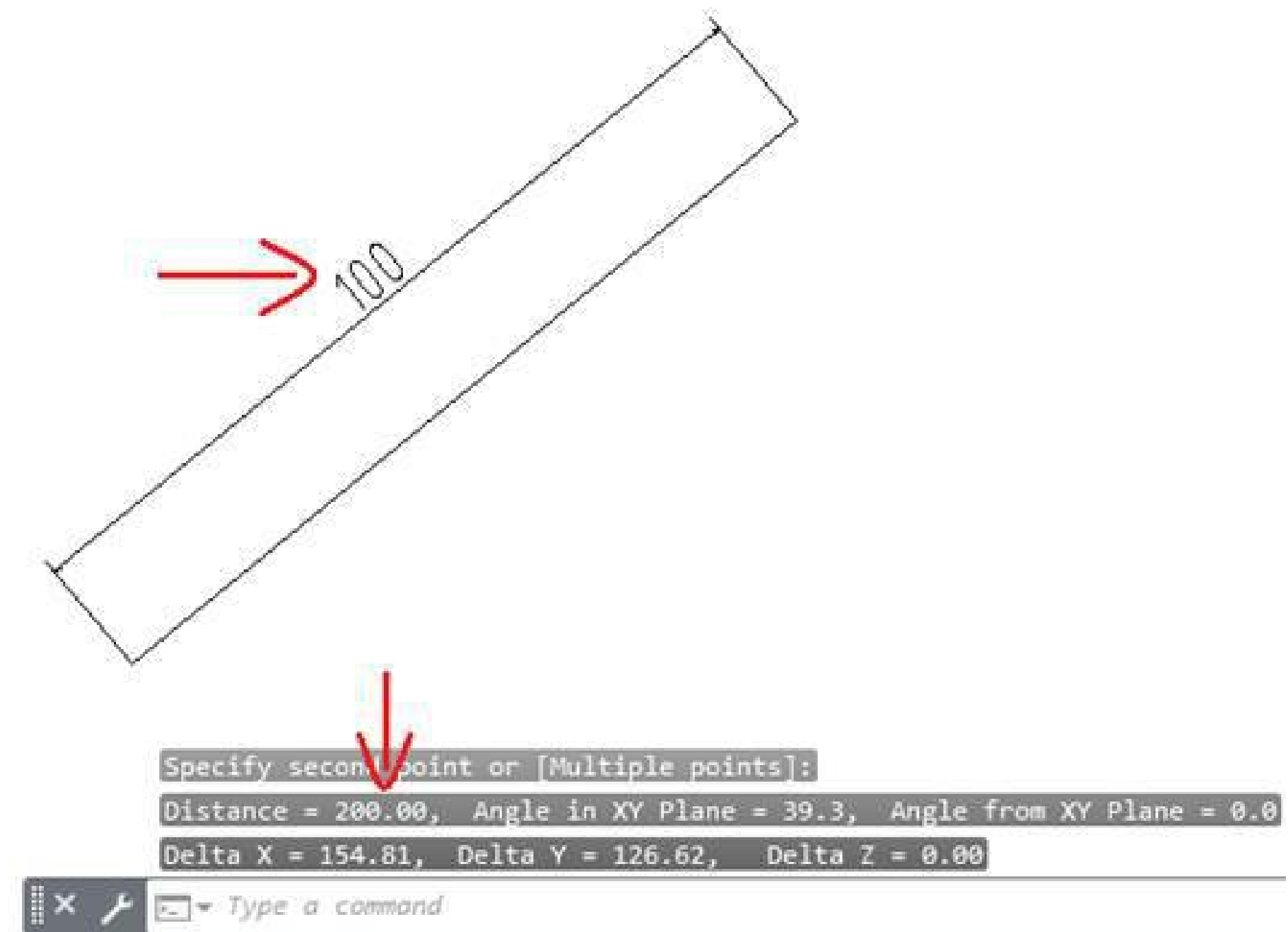
Actual value (**A**) = **Distance** = display size value (**D**) = 100



Case 2: The drawing person is drawing with a Real draw scale = 2:1 = 2

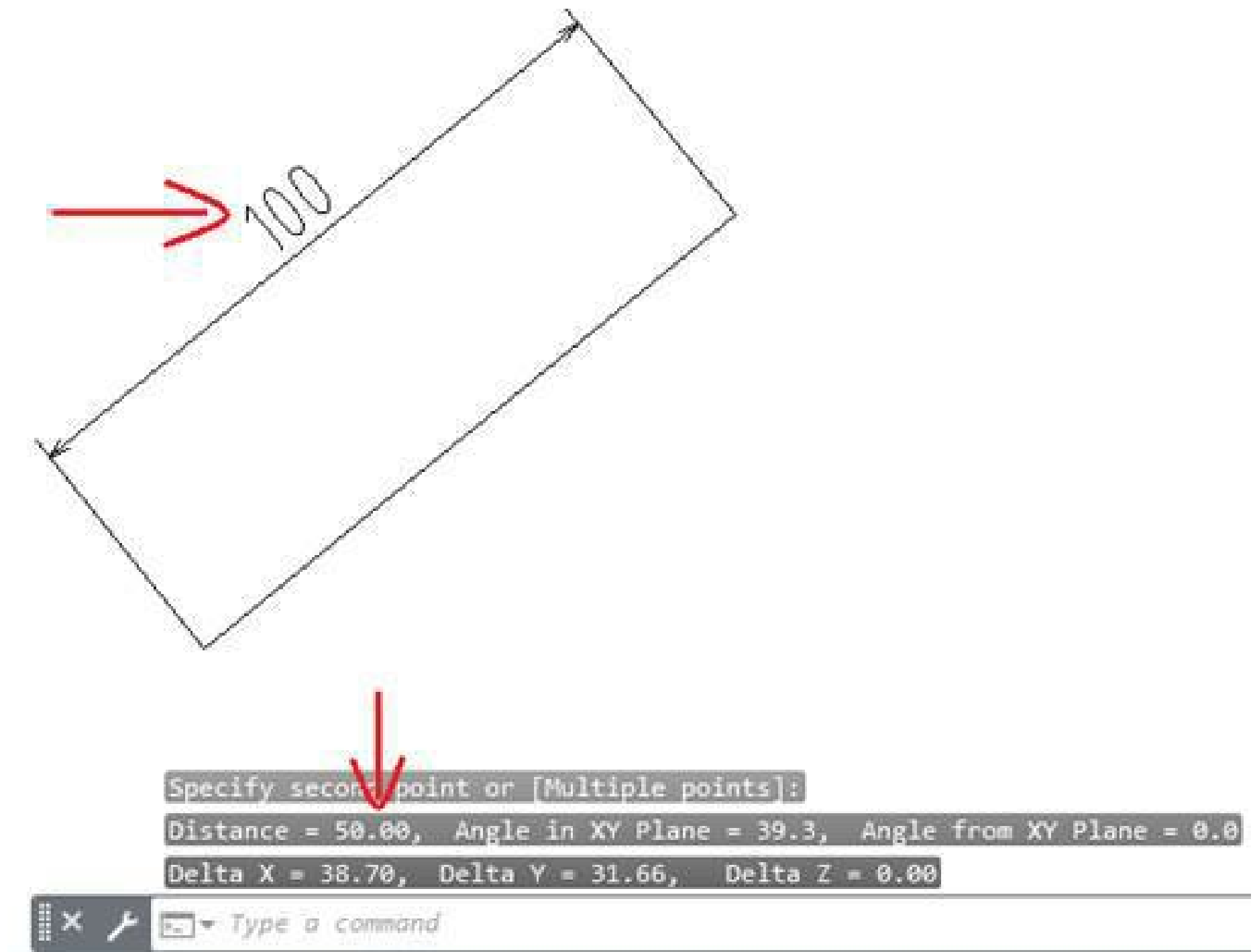
The actual value Distance (A) = 200, but they write the display size as 100 (D)

This expression you will always see in the Detail section.



Case 3: The drawing person is drawing with a Real draw scale = $1:2 = 0.5$

The actual value Distance (A) = 50, but they write the display size as 100 (D)



(Case 3 will be very rare)

The drawing cases are different from the 1:1 Real draw scale, people often intentionally make it different to scale the drawing after printing (they use the way to edit the Scale factor), but here is not the right way to do it and you don't have to care about it; The second reason could be because they intentionally EDIT the size, and I don't recommend this!

We have a formula:

$$\text{Real draw scale} = \frac{A}{D} = \frac{\text{Actual}}{\text{Display}} = \frac{\text{Actual dimension value entered by the drawing person (measured with the DIST command)}}{\text{Dimension value that the drawing person wants to display on the screen for the viewer (when measured by DU, DAL, ...)}} (*)$$

- Real draw scale = 1: we say that they draw it to the real dimension
- Real draw scale > 1: Enlarged drawing ratio (usually used with detail section)
- Real draw scale < 1: Zoom out Real draw scale (not used because of lack of reality)

The principle of technical drawing on Model space should always be to Real draw scale = 1:1.

Zoom scale is used in the extracted images to zoom in locally at certain locations of the object. Therefore, we often see the Real draw scale of the detailed extract as 2:1, 4:1, 10:1, etc. The Real draw scale of the detailed extract is also the Real draw scale of the objects inside the extracted image.

Let's go back to the three examples above:

- Case 1: A = D = 100 => Real draw scale = 1:1
- Case 2: A = 200, D = 100 => Real draw scale = 2:1 = 2
- Case 3: A = 50, D = 100 => Real draw scale = 1:2 = 0.5

Thus, understanding the Real draw scale helps you to assess whether the drawing of another person you are viewing is drawn correctly with the real 1:1 scale or not? If it is not a 1:1 Real draw scale, check it seriously to avoid errors

with real objects.

Therefore, I recommend you on the Model space: Always draw everything with Real draw scale = 1:1 (except with detail section)

You already remember how to check the actual size of a line segment, just use the DIST command!

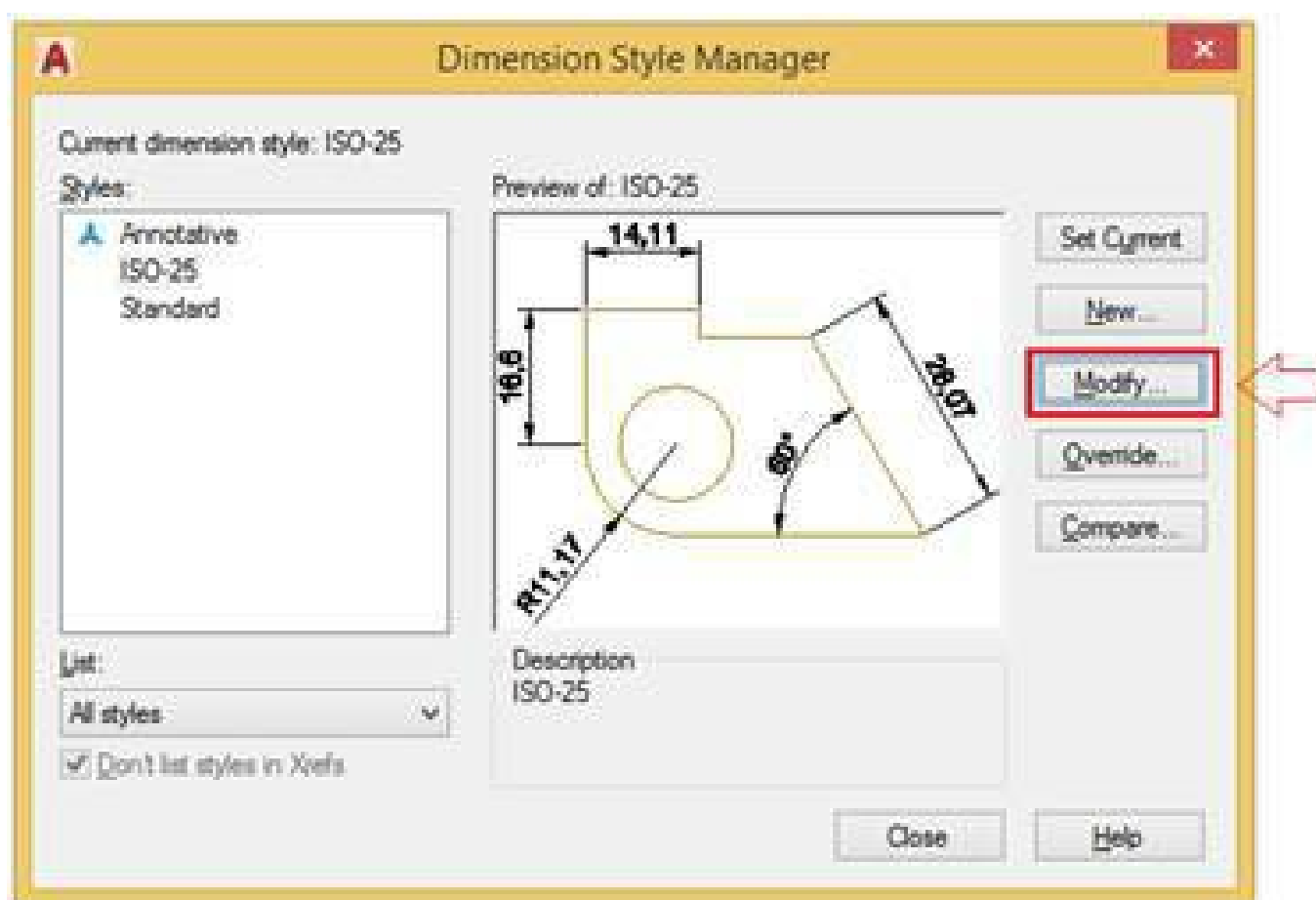
Rest assured that the DIST command will always give the correct result, no matter how much you adjust the Scale factor. So it's believable!

Now you know one more tool to help you become wiser.

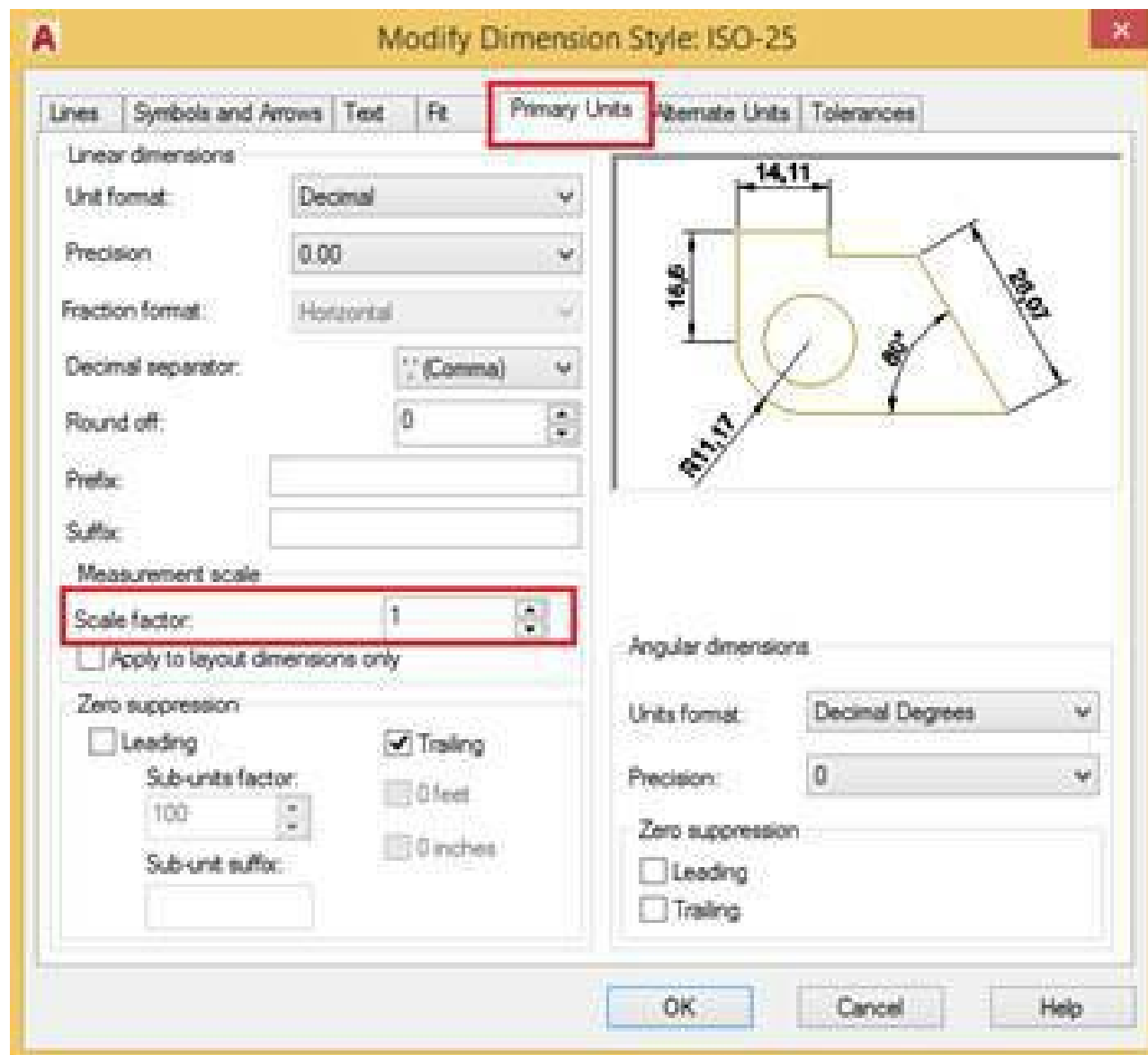
Question: Is there a way to make sure I always draw with a Real draw scale = 1:1?

AutoCAD has always left you by default to draw with the drawing ratio = 1:1, so you don't realize it. To make sure you always draw with a real draw scale = 1:1 you do the following:

- Before proceeding to draw anything, type the command D ↵ (↵ means the Enter command)
- Select the **Dimstyle** that you intend to use for dimensioning, click **Modify**



- Select **Primary Units**
- Look at the *Scale factor*. By default, it is always 1 so you draw the correct Real draw scale = 1:1



After checking that the *Scale factor* is exactly 1, you can safely start drawing.

If it's not 1, double-check it before returning it to 1.

The way I currently do it is to put this setting into a permanent Template file and then I don't have to think about it anymore. By default, I always draw with a Real draw scale = 1:1.

Question:

Is there a way to know what Real draw scale someone is drawing with? I suspect they don't draw to a 1:1 ratio.

Of course, there is a way!

- The first way is to check the Scale factor in the Primary Units tab as I explained above.

$$\text{Real Draw scale} = \frac{1}{\text{Scale factor of Dimstyle that is used to note that dimension (**)}}$$

- The second way: type the command **DIMLFAC**✓.

The value of DIMLFAC (it is always 1 by default) is displayed as equal to the Scale factor of the current Dimstyle. Of course, these two coefficients (Dimlfac and Scale factor) always come in pairs with the same value as the current Dimstyle.

2. Drawing scale

Drawing scale is a very basic concept in technical drawing, but not everyone understands what it means.

I would like to confirm that the drawing scale is only meaningful when the drawing is printed on paper. There is no concept of drawing scale on model space (Model) because on Model is infinite space, by default it must be drawn in 1:1 "Real draw scale" (except for the detail section).

Therefore, for the drawing to be printed on paper, and it has the correct scale, on the title block of the drawing on the Model (or Layout) you must write down the value of the Drawing scale that you intend to print based on the size paper you have selected.

So in summary, what is the drawing scale?

The example below will help you better understand the nature of the “Drawing Scale” and why it exists?

How do you show the vertical projection of a house on an A3 sheet of paper?

To show the vertical projection of that house, they need to hold an A3 sheet of paper (420x297mm) to stand in front of the house and enlarge the paper until it is enough to cover the front of the house. Let's say, the zoom factor is 50x (increasing both dimensions by 50x).

Project the front of the house onto that giant sheet of paper. We have a giant drawing.

However, the reality is that there are no paper sizes and printers of such enormous size. We will print the facade of the house by reducing the entire drawing on that giant sheet of paper to the size of just 1 sheet of A3 paper (420x297mm). On that A3 sheet of paper, the actual size of the house is reduced by 50 times, but the size shown does not change.

It means that the facade is 10 meters, then that paragraph on A3 paper is also noted 10 meters. But, if you use a ruler to measure that 10 meters on paper and 10 meters in real life, it's completely different, the size of 10 meters in real life is reduced by 50 times compared to when measured on paper (using a ruler to measure on paper is only $10000 / 50 = 200\text{mm}$). And then, we say this drawing has a scale of 1:50 (Drawing scale = 1:50)

Since then, the concept of the “Drawing Scale” was born. In this case, 1:50 is the scale of the miniature drawing.

What about Model space?

Remember, the concept of the "Drawing Scale" was born before people invented graphic software.

In the example above, the A3 sheet of paper with a real-life name frame has been imagined to be magnified 50x. Whatever you do in real life, the software will do the same. On the Model space, the vertical projection of the house is kept at its true size with a 1:1 Real draw scale, the A3 frame (420x297mm) is enlarged 50 times to become an A3' size ($A3' = A3 \times 50$). When printed on paper, the A3' frame is again shrunk 50 times to a real A3 sheet and we get a 1:50 scale drawing (or write 1/50)

The same interpretation applies to the scale of the enlarged drawing (do the opposite).

And the same for all other paper sizes, from A0 -> A4.

There is a definition of Drawing Scale as follows:

Drawing scale is the ratio between the size measured on the drawing after printing on paper and the corresponding size measured in reality.

Eg: On a map with a scale of 1/1000, we understand: 1 unit on the map corresponds to 1000 units in reality.

Architectural drawings of 1/50 scale, 1 mm on that drawing after printing on paper corresponds to 50 mm in reality.

OK! Do you now understand the concept of drawing scale?

Types of drawing scale:

There are 3 types of drawing scale: intact, enlarged, and reduced

- Original aspect ratio: 1:1

(meaning the size of the object when measured on paper (after printing the drawing) is the same as the actual size when measured on the same object in real life, no matter what kind of paper it is)

- Reduced drawing scale: consists of 1:2, 1:2.5, 1:4, 1:5, 1:10, 1:15, 1:20, 1:25, 1:40, 1:50, 1:75, 1:100, 1:200, 1:400, 1:500, 1:800, 1:1000

(applies to parts, large objects; almost all construction drawings use drawings with reduced scale)

- Enlarged drawing scale: 2:1, 2.5:1, 4:1, 5:1, 10:1, 20:1, 40:1, 50:1, 100:1

(Applicable to parts, small-sized objects, or the scale of the Detail section. Enlarged scale drawings are often used in mechanical drawings)

However, depending on the layout of the printed page, the aesthetic eye, the balance of the overall drawing, it is possible to choose a scale according to a different number, this is not wrong but should be limited.

What is 1:2, 1:10, 5:1 drawing scale?

- 1:2 is the scale of the miniature drawing. The actual size is twice the size measured on the paper.
- 1:10 is the scale of the miniature drawing. The actual size is 10 times bigger than the size measured on the paper.
- 5:1 is the scale of the enlarged drawing. The size measured on paper is 5 times larger than the actual size

3. Scale of the Detail section

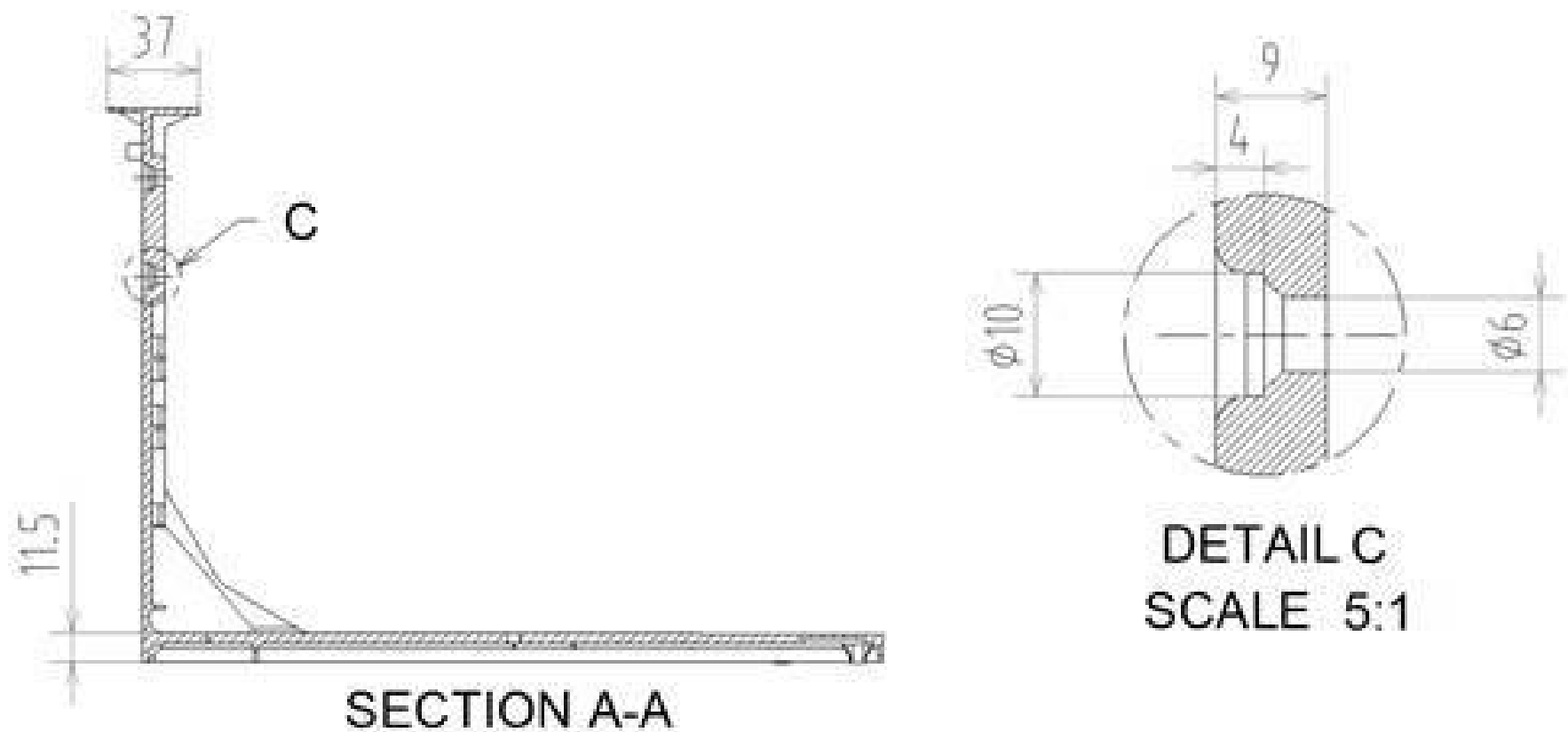
The detail section is a locally enlarged image of an object's location to clarify the shape and structure at that location, but the size remains unchanged compared to the original true size at that position of the object. The **Detail section** should clearly state the Real draw scale.

1 Detail section includes: magnified part of object, actual size, scale of extract and can add notes

The real draw scale is the zoom level of the extracted image compared to the original image, which is also the local drawing ratio at that position. Example: 2:1, 4:1, 5:1, 10:1

The real draw scale is written just below the **Detail section**.

An example of the Detail section:



Chapter 3: Title Block And Principles For Building Drawing Scale

1. The basic requirements of a Title Block

Size of **Title Block** (with real draw scale 1:1): A3 - 420x297mm, A4 – 297x210mm

For the **Title Block**, each company/unit will have its presentation specification, along with the company's brand name/logo. In my opinion, the mechanical **Title Block** will also include the following 4 main parts:

- The frame that covers the drawing
- The content of the name frame (name details, materials, designers, ...)
- Part history of drawing changes (Revision)
- Tolerance lookup table (optional)

However, some companies print A4 size, so to save space for drawing presentation, they allow to omit the tolerance table. Tolerance tables are issued separately and readers refer to them separately.

				<table><tr><td>Num</td><td>PIC</td><td>Date of change</td><td>Content changes</td></tr><tr><td>1</td><td></td><td></td><td></td></tr><tr><td>2</td><td></td><td></td><td></td></tr><tr><td>3</td><td></td><td></td><td></td></tr></table>				Num	PIC	Date of change	Content changes	1				2				3			
Num	PIC	Date of change	Content changes																				
1																							
2																							
3																							
TECHNICAL REQUIREMENTS: 1. Color 2. Appearance (bevel glass, warping) 3. Hardness : with metal 4. Surface treatment (MFZn2-Cr, 55, 72h)																							
Title	Name	Sign	Date	Num	Part Name	Drawing number	Ver	W/g	Qty	Material	Note	DWG											
Design							00		1			0											
Check				MODEL NAME		Scale	1:1	HYDROELECTRIC PROJECT JOINT STOCK COMPANY															
Approve						Projection																	

2.3. Basic principles for building drawing scale

Principle 1:

On Model space, “display dimension” should always be noted as exactly the “true dimension” of the object, with a real draw scale of 1:1

This means that whether you do a 2D design yourself in AutoCAD or the data is a projection exported from another 3D software, the displayed dimension of the object in Model space is always the actual dimension corresponding to the real draw scale 1:1.

For example, a pencil in real life is 150mm long, on the Model it must also satisfy 2 conditions:

- The displayed dimension value is true to the actual dimension, equal to 150mm
- Make sure the **Real draw scale** = 1:1 (when measuring on the Model space by the DIST command, the correct result is 150mm)



Actual dimension



Consequence:

When the object is too large (a house, a car, ...) it will not be possible to place it on an A3, A4, or A0 sheet of paper. They were forced to shrink it so that it could still be fully displayed on a piece of paper.

From there, the concept of drawing scale was born:

From that fact it can be inferred that:

- If the drawing scale is 1:n, then the title block on the Model has been enlarged n times
- If the drawing scale is m:1, the title block on the Model has been reduced by m times (factor equals 1/m).

Principle 2:

In Model space, whatever the drawing scale, keep the shape and scale the title block.

For data that is your projection, drawn by yourself in AutoCAD or exported from 3D software, keep it with the correct real draw scale 1:1.

You will scale the title block by a certain factor α .

- If $0 < \alpha < 1$: you have a drawing with the enlarged drawing scale
- If $\alpha > 1$: you have a drawing with the reduced drawing scale
- If $\alpha = 1$: you have the original drawing, drawing scale = 1:1

Principle 3:

On the same paper size, all drawings, regardless of the drawing scale, must be the same basic height of the text.

The basic height is the height of most of the text on the drawing, including Technical requirements, digits representing the value of the dimension, the content of notes, content in the title block. In the title block, some items may allow text with a height greater than the basic height.

- The basic height of text on A4 size is usually 2.5mm or 2mm
- The basic height of letters on A3 size is usually 3.5mm

Refer to the table below for the specifications of the font sizes used on the drawings below.

Num	Content	Font	Color	Text height (mm)				
				A4	A3	A2	A1	A0
1	Technical requirements	Arial	White	2.5	3.5	5	7	10
2	Notes, information	Arial	White	2.5	3.5	5	7	10
3	Digits used to note the dimension	Iso, Block font	Purple	2.5	3.5	5	7	10
4	Digits used to note the ordinal number of the part	Iso, Block font	Red	2.5	3.5	5	7	10

Consequences of rule number 3?

- ✓ Drawing scale is equal to 1:n, then the height of text in the drawing frame on the Model space needs to be Scale n times (enlarge the text height n times)
- ✓ Drawing scale is equal to m:1, then the height of text in the drawing frame on the Model space needs to be scaled 1/m times (reduce the text height m times)

As follows:

With a drawing on A4 size, assuming we specify the basic height of the letter after printing to be 2.5mm, the height of the letter (H) on the Model will be:

- Drawing scale = 1:1 => $H = 2.5 \times 1 = 2.5 \text{ mm}$
- Drawing scale = 1:2 => $H = 2.5 \times 2 = 5 \text{ mm}$
- Drawing scale = 1:10 => $H = 2.5 \times 10 = 25 \text{ mm}$
- Drawing scale = 2:1 => $H = 2.5 \times (1/2) = 1.25 \text{ mm}$
- Drawing scale = 4:1 => $H = 2.5 \times (1/4) = 0.625 \text{ mm}$

With a drawing on A3 size, assuming we specify the basic height of the letter after printing to be 3.5mm, the height of the letter (H) on the Model will be:

- Drawing scale = 1:1 => $H = 3.5 \times 1 = 3.5 \text{ mm}$
- Drawing scale = 1:2 => $H = 3.5 \times 2 = 7 \text{ mm}$
- Drawing scale = 1:10 => $H = 3.5 \times 10 = 35 \text{ mm}$
- Drawing scale = 2:1 => $H = 3.5 \times (1/2) = 1.75 \text{ mm}$
- Drawing scale = 4:1 => $H = 3.5 \times (1/4) = 0.875 \text{ mm}$

Thus, with the text height on the drawing, we have the formula:

Text height on drawing in Model space
= Basic height of text in paper size x (Drawing scale)⁻¹

Did you get it!

Chapter 4: You Need A Template File

What is a template file?

The AutoCAD Template file is in .dwt format.

Every time you press Ctrl N to open a new file, you will see a dialog box appear asking you to choose which file?

That is the list of Template files that AutoCAD has prepared for you. Of course, it's made for millions of users, so it's nothing special about your work. It will be better if you create your Template file just for you.

The template file is the sophisticated toolkit of magicians, the sharp ax that you have prepared every time you enter the forest. Wherever I work, I have a habit of building a Template file suitable for the working environment there. Because each place will often have a different nature of work, the expression of the name frame, the conventions of the contours, the standards may also be different and may arise new requirements. Therefore, flexibility is required.

Building a Template file can take you a long time, depending on how detailed you are, and also depending on your "creativity". It could be a week or even months because sometimes you throw something in the Template file.

The benefit of the Template file is that you will only build it once and forget about it, when you need to use it, just use it, without thinking. In terms of investment language, creating a Template file is like you are just “investing once and profiting forever”.

Well worth the effort to build it!

Later in the book, I will provide detailed instructions to help you build a complete Template file, what to include, how to install it. Surely you will create yourself a very sophisticated and classy Template file.

Interface Settings And Optimal Settings

First, I recommend that you choose a version of AutoCAD that “fits” your computer's configuration. Of course, the higher CAD generations, the number of tools will also increase, but you will probably never use them.

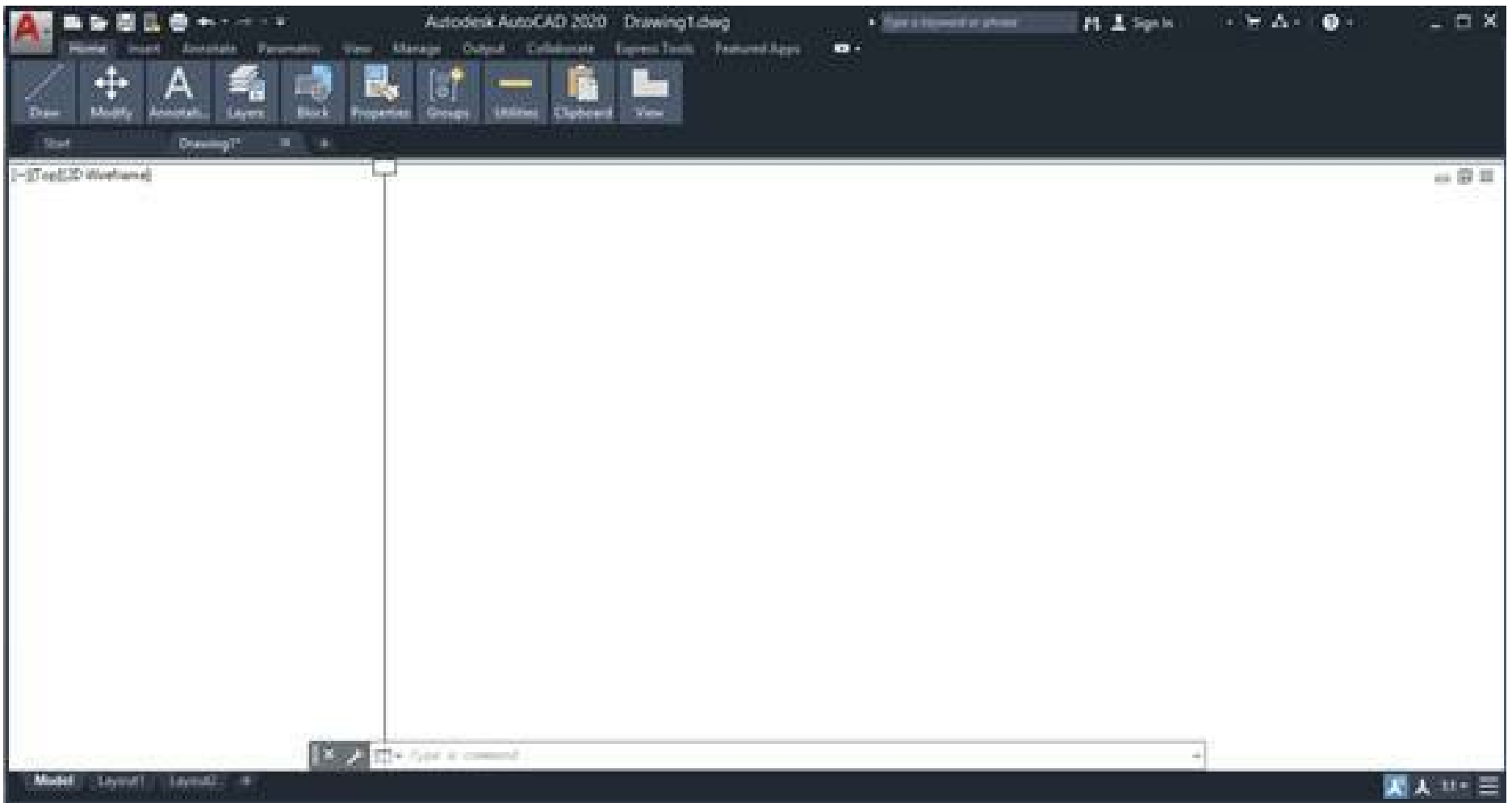
In my opinion, AutoCAD from 2010 and up is quite good. I have been using CAD 2012 version for many years because it is suitable for my machine configuration (core I5, RAM 4GB, Card 1GB, HDD) but I can still guide others on CAD 2015, 2018, 2020, 2021 without any difficulty!

With all versions of AutoCAD that have gone through more or less, I have remained faithful to the 1 interface style and 1 Options setting. I find that those settings are optimal, giving me the most spacious workspace, fastest processing of work.

As I said, I'm not imposing, you may or may not like it, but I share what I think is the best because I'm doing the same.

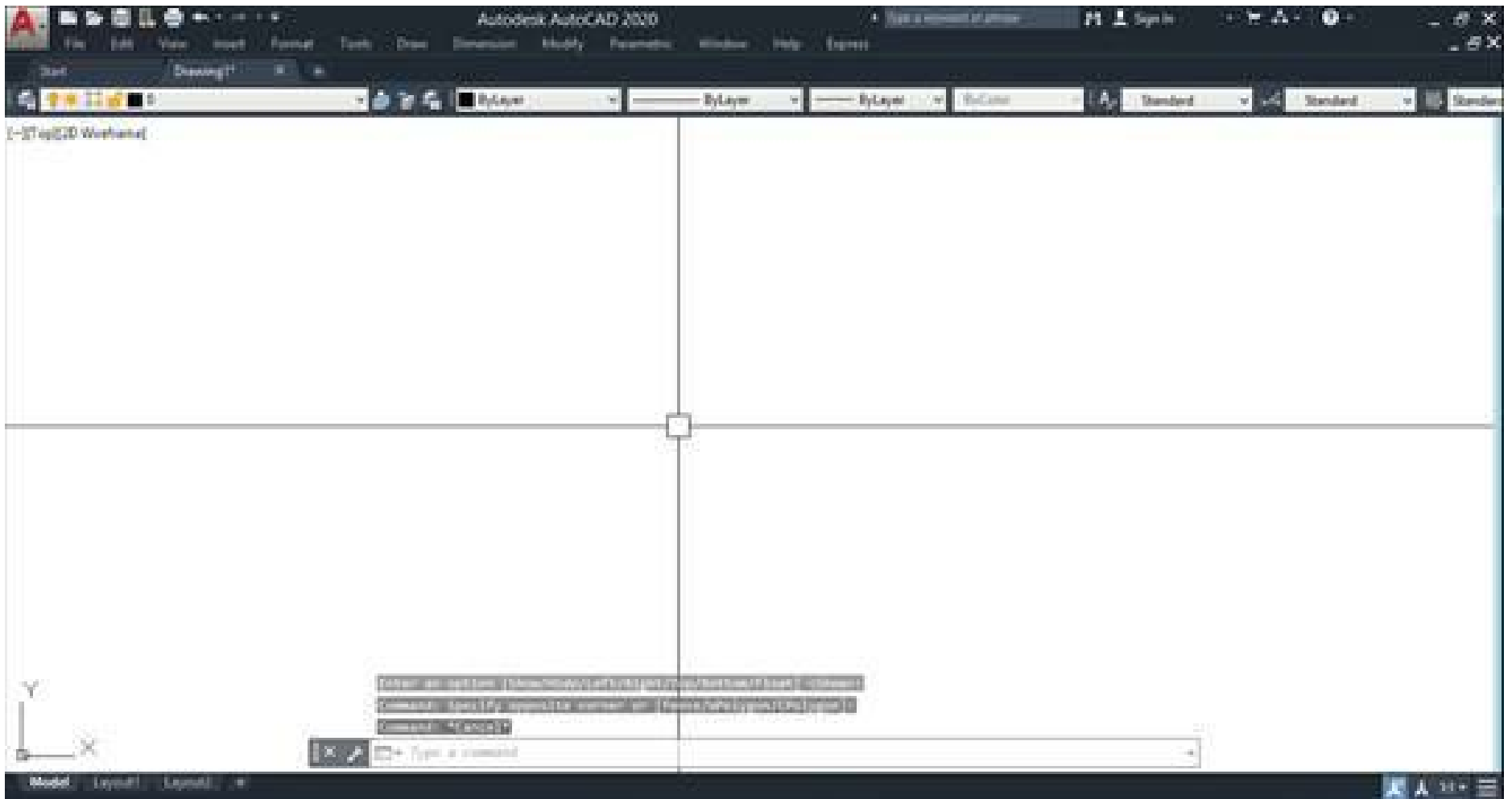
1. Classic is better than Ribbon

With high-end CAD versions, the default interface after installing the software is the RIBBON interface. In my opinion, it is only suitable for those who are new to AutoCAD because it shows all the commands on the screen. Therefore, it takes up a lot of display space, and that would not be an appropriate option to work with.



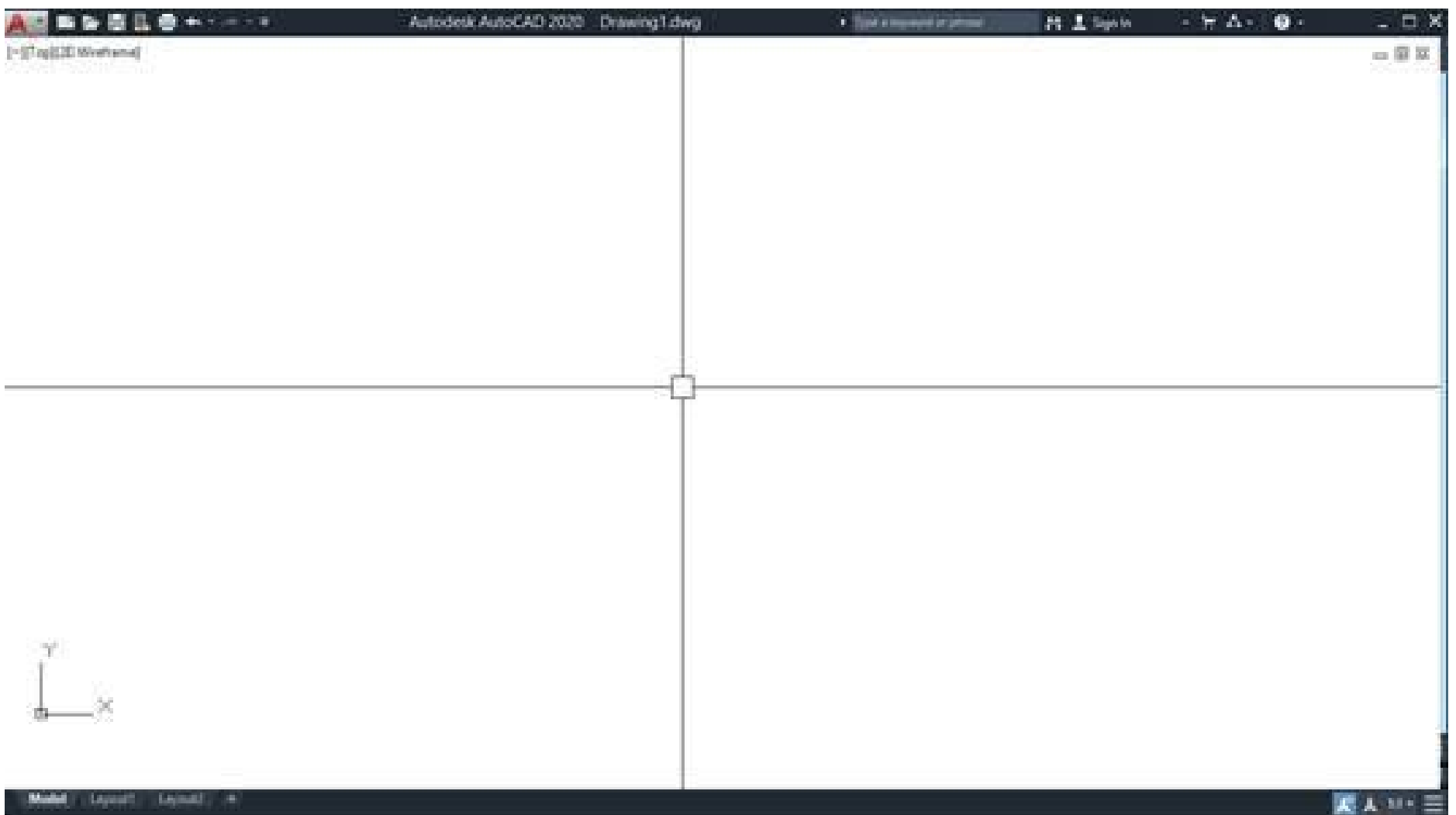
AutoCAD 2020 RIBBON interface (takes up about 25% of the screen space)

The classic interface gives you more space. And of course, you have to memorize the shortcut commands.



AutoCAD 2020 Classic interface (takes about 15% of the screen space)

And you can even optimize to use almost 100% of the screen space if you want.



I'm using the 2nd method.

OK! Now I want us to agree that we will only use the Classic interface for work, consider this a win-win deal for you. Just follow me and don't worry.

How to make the Classic interface work:

- **Step 1:** Type **RIBBONCLOSE** ✓ to close the Ribbon
- **Step 2:** Type **MENUBAR** ✓
 - At the prompt "Enter a new value for *MENUBAR*": enter a value = 1 ✓ to agree to display the Menubar on the screen.
 - (If you type the value = 0, it means you will hide the Menubar)
- **Step 3:** Show 3 optimized toolbars: Layers, Properties, Styles

To show the Layers toolbar, do the following:

- Type the command: **-TOOLBAR** ✓



- At the prompt – *TOOLBAR Enter toolbar name*, you type: layers ✓



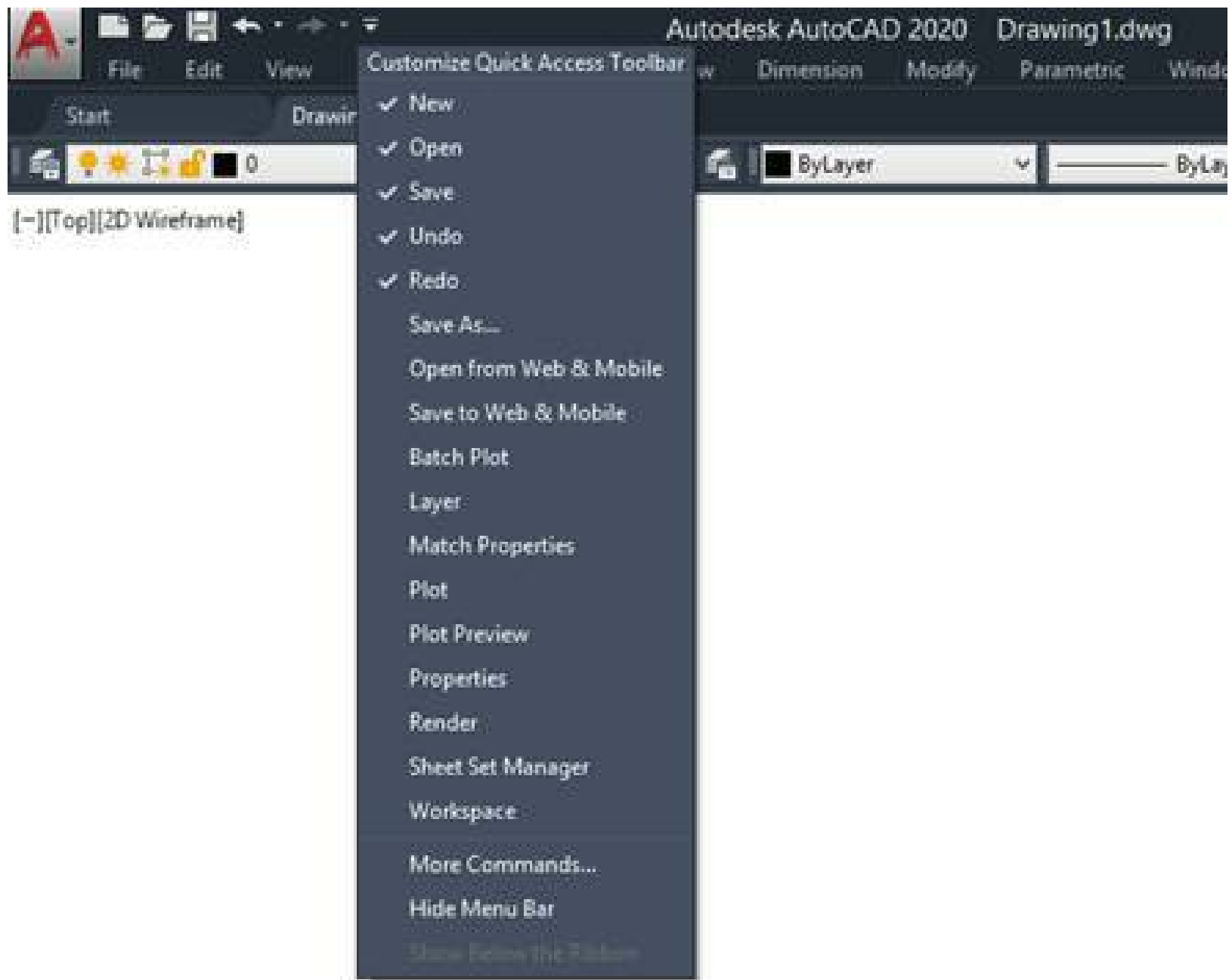
- Continue typing ✓ one more time to select the Show option to show the Layers toolbar



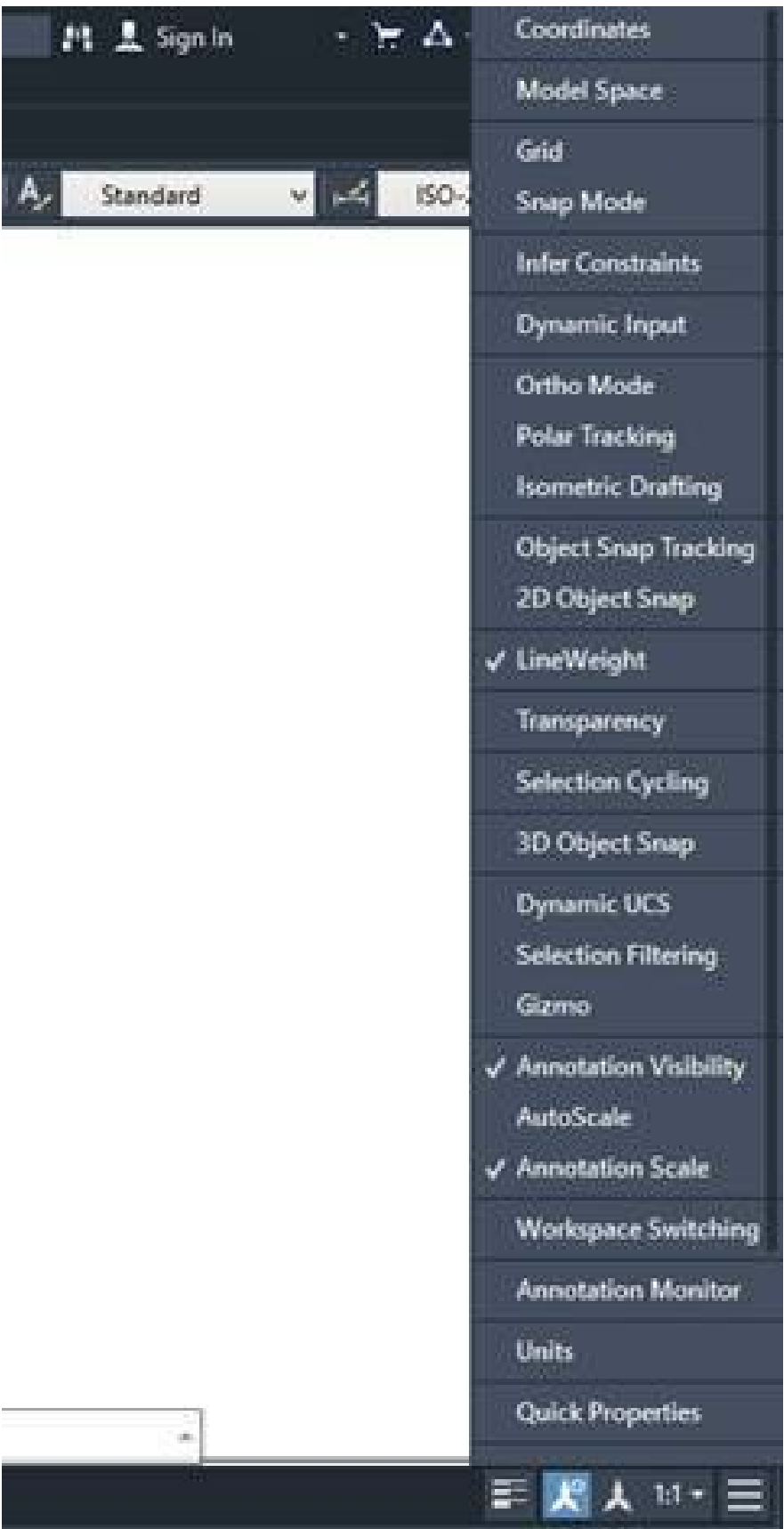
With 2 toolbars Properties and Styles, we will do the same to show them up.

- **Step 4:** Should optimize some more quick options.

Click to bring up some options: New, Open, Save, Undo and Redo because you can use them a lot.

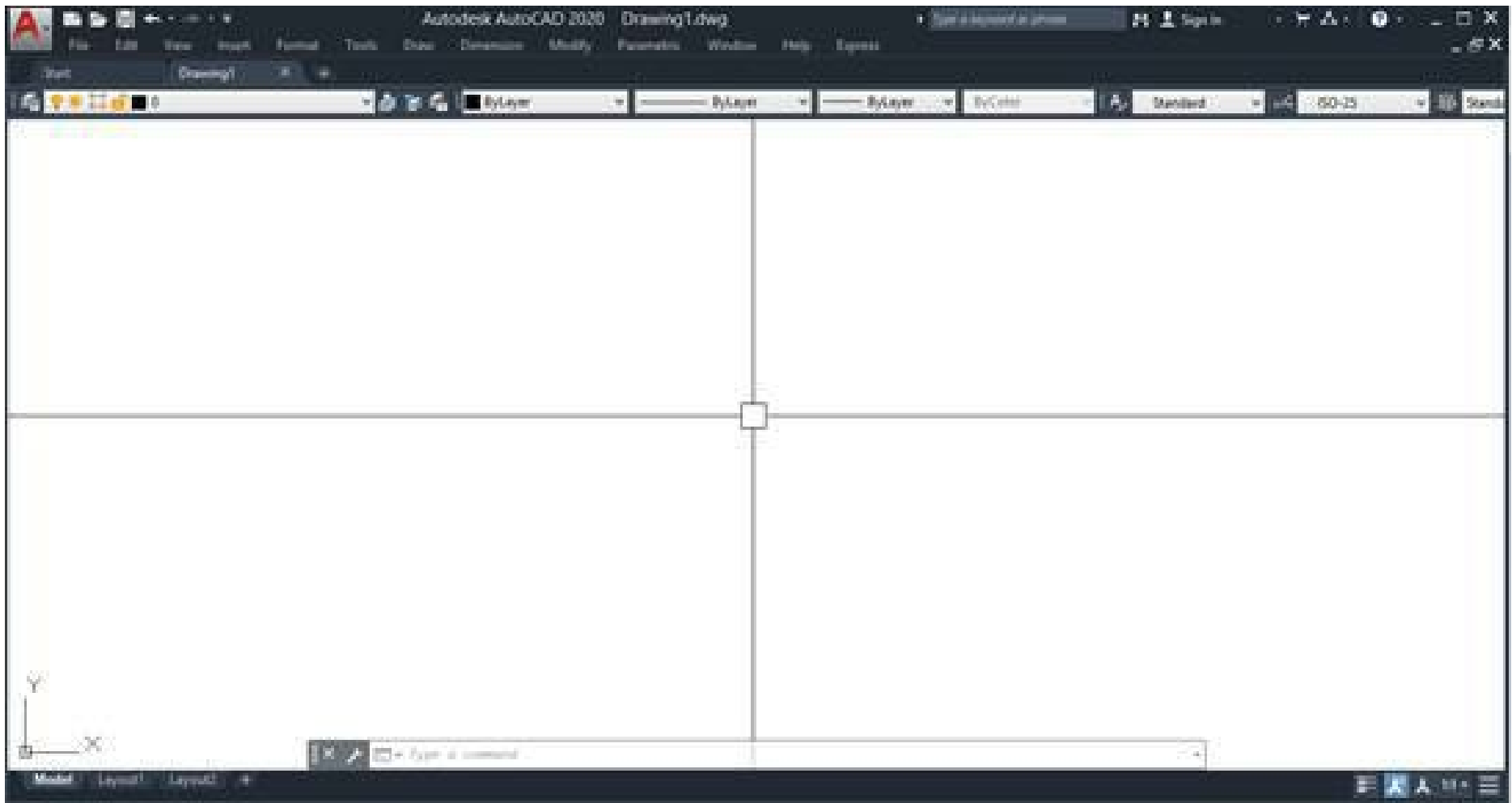


- Step 5: Show only a few options in the lower right corner of the screen. Hide less because it's not in use, so it's a lot of eye pain. I recommend you keep only the Annotation Visibility, Annotation Scale, and Lineweight.



It's okay! Now review your results.

Simply like this!



Note: *If you want to restore the Ribbon interface, type the command **RIBBON**✓*

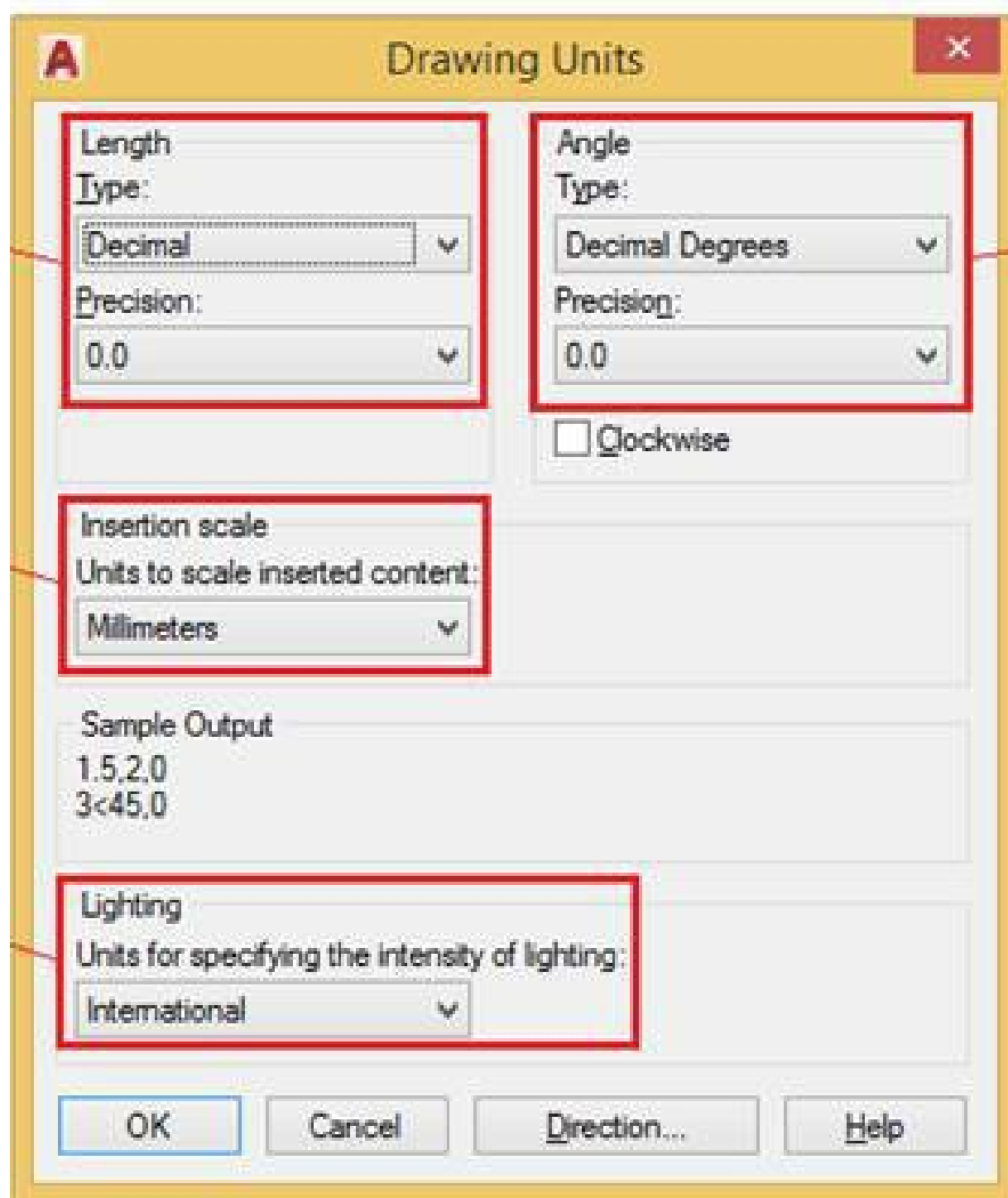
And if you want to clean the entire screen, without any toolbars, press the key combination at the same time:

- Ctrl 0: to show/hide all 3 toolbars above
- Ctrl 9: to show/hide the Commandline input bar

OK. You are very well prepared.

2. Drawing unit

- Type **UN**✓ (UNITS) to enter the Drawing Units dialog box and set the following:



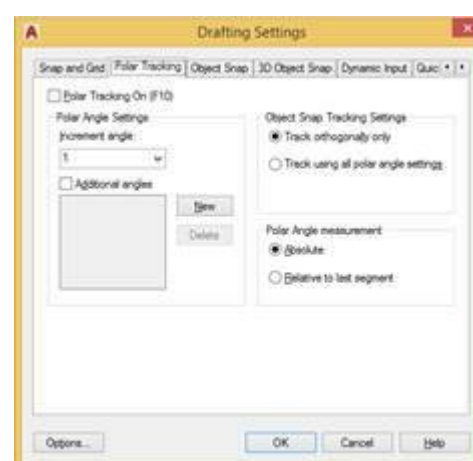
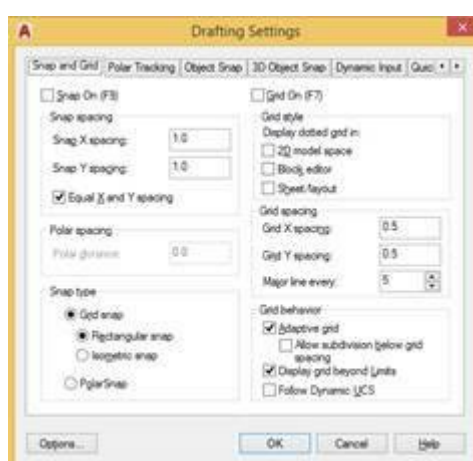
I will explain what is important and necessary.

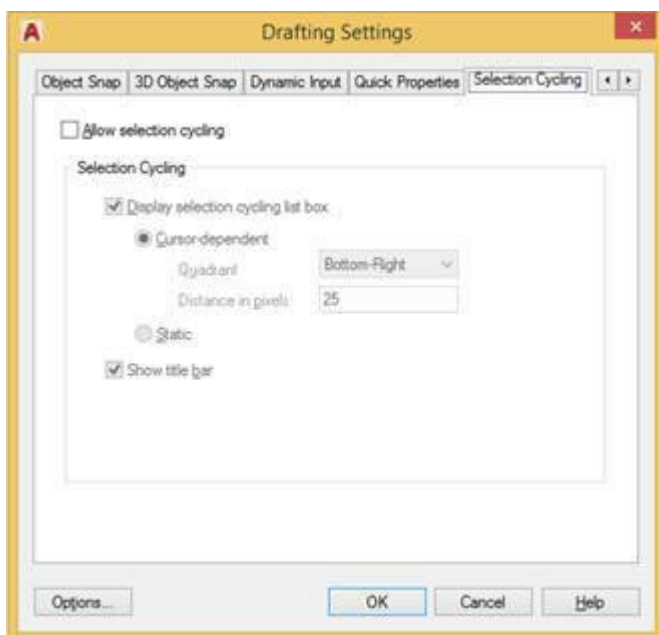
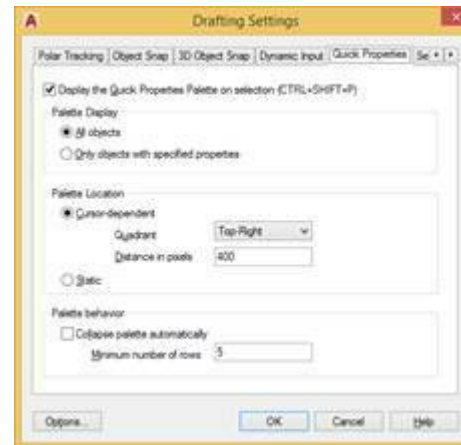
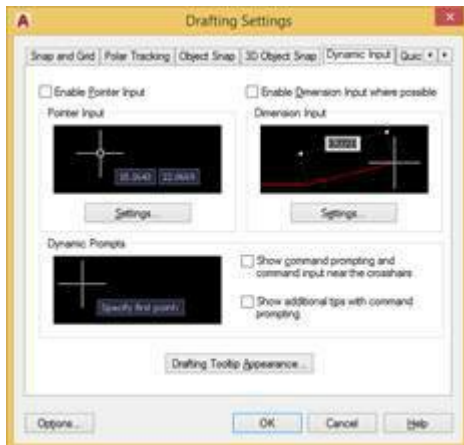
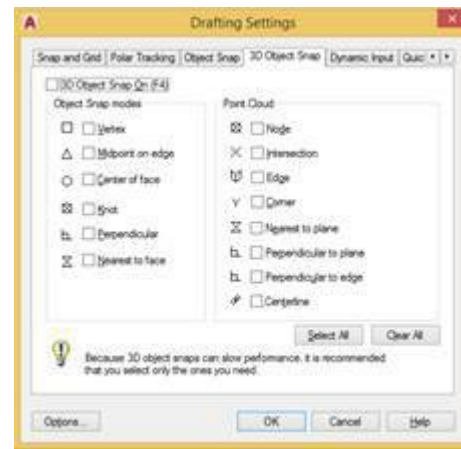
3. Draft Settings

To access the Draft Settings dialog box, type the command: **DS**✓.

Draft Settings contains settings for display mode, snapping, cursor jumps, quick properties display, and more.

You install the following:





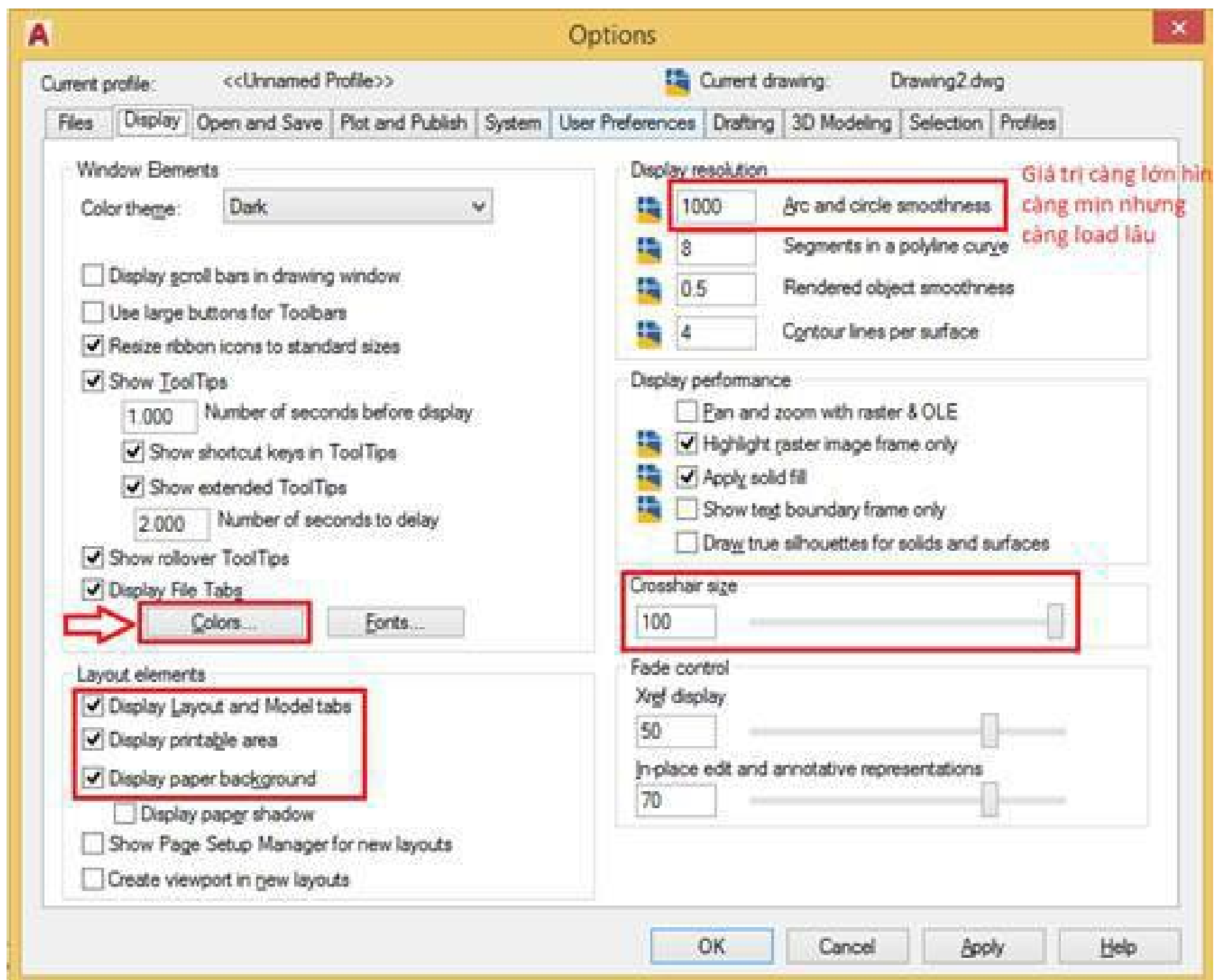
In Drafting Settings, there are 5 important keys you should keep in mind:

- F3: enable/disable the tracking mode
- F8: enable/disable perpendicular drawing mode
- F7: enable/disable point grid mode
- F10: enable/disable drawing mode with angle orientation
- Shift Ctrl P: enable/disable the quick properties dialog (Quick Properties)

4. Options

Options contain nearly every setting of AutoCAD. This is a very important dialog box, you must master Options. However, in this book, I will focus only on the important content without extensive instruction.

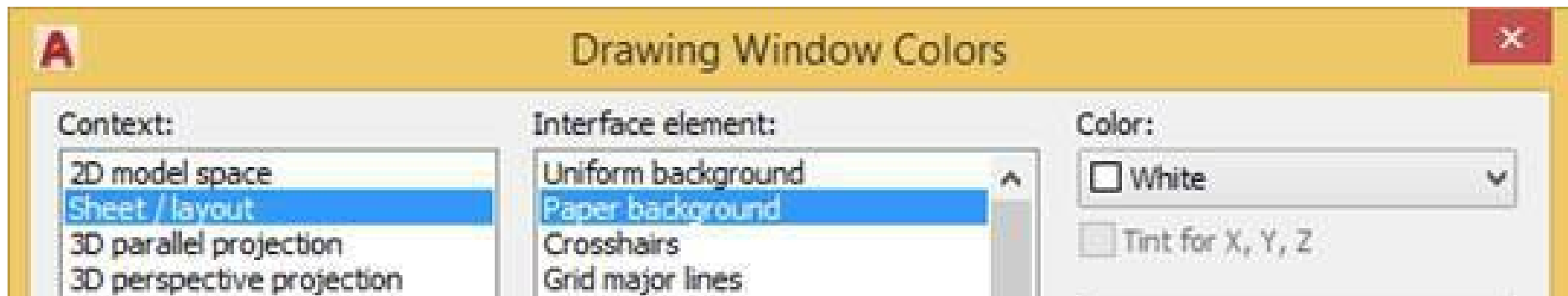
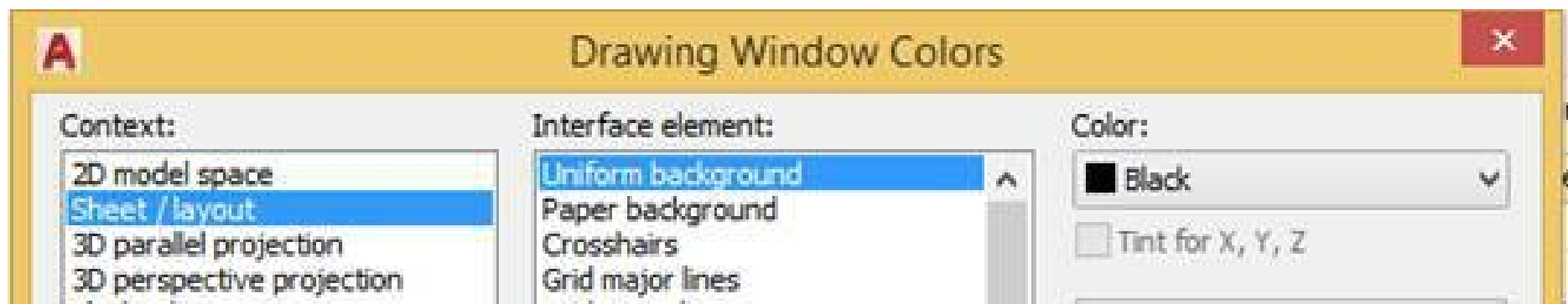
4.1. Display - Focus on Color



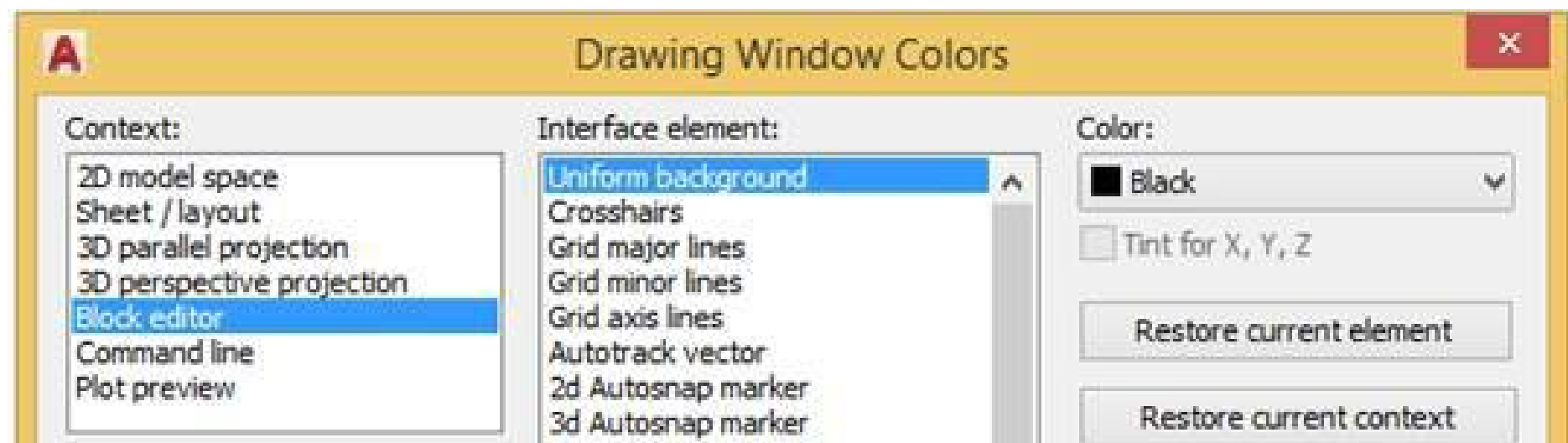
Color Settings: The Model Environment is Black



- Sheet/layout environment:

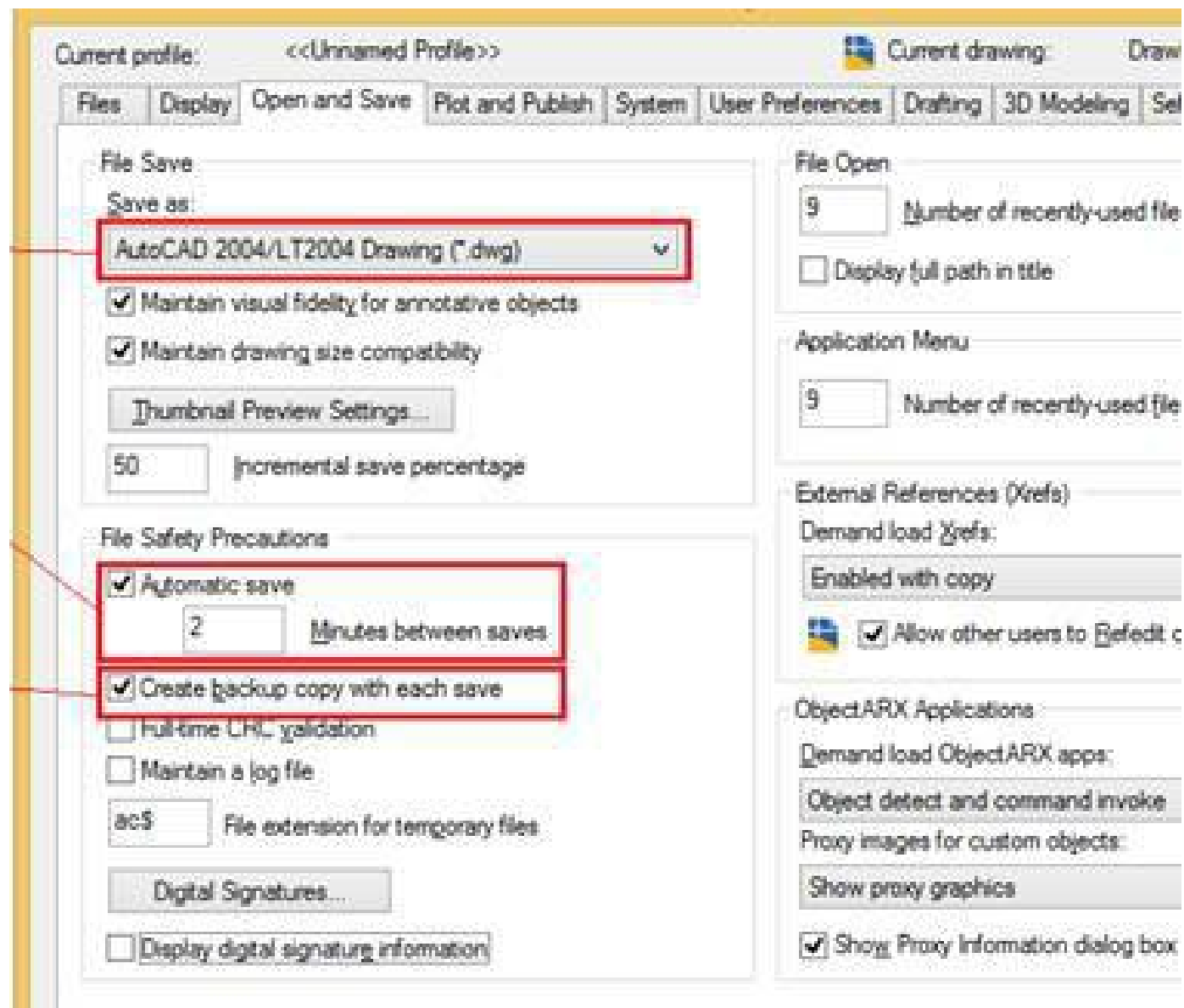


- Block Editing Environment

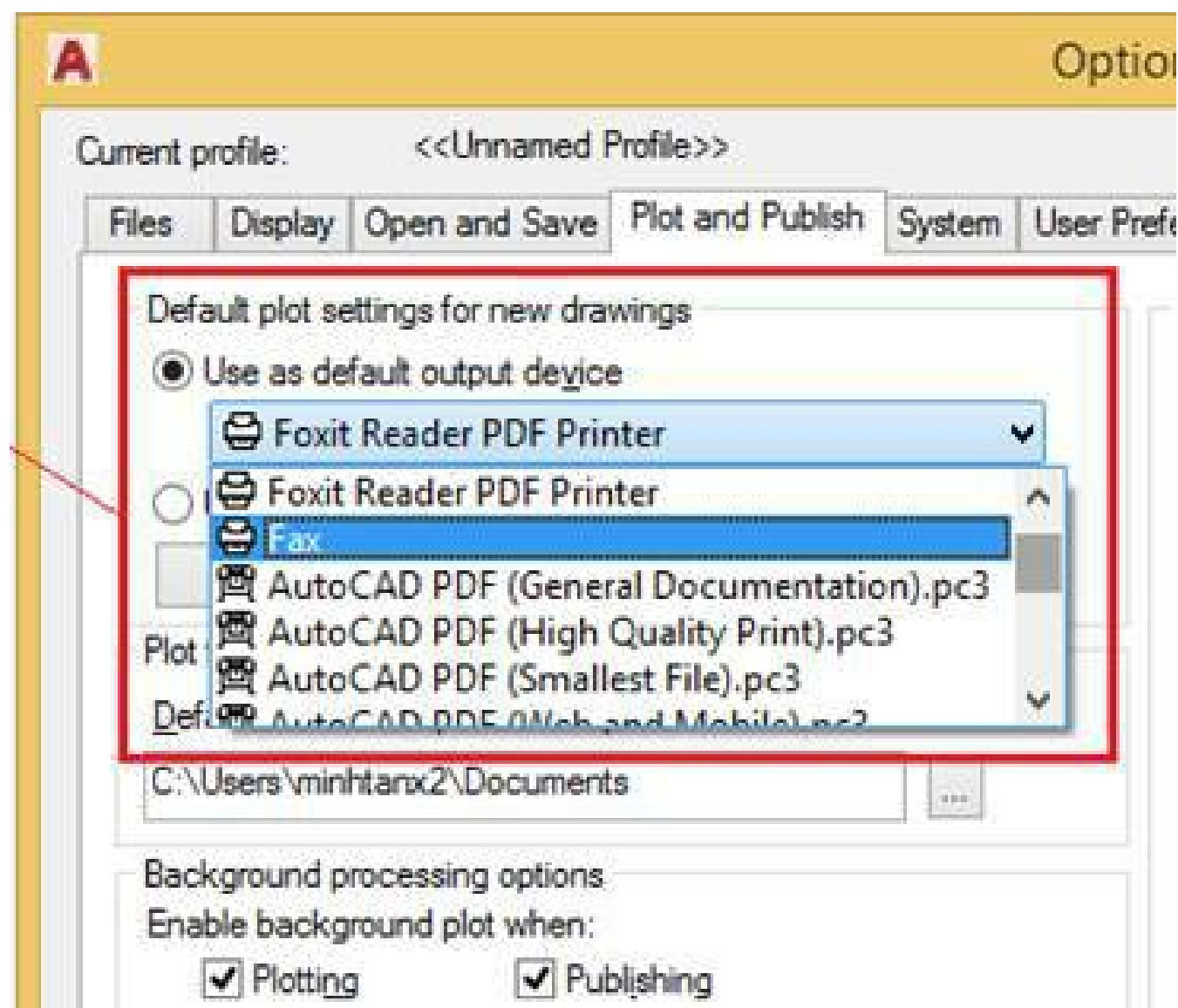


4.2. Open and Save

- The most important settings

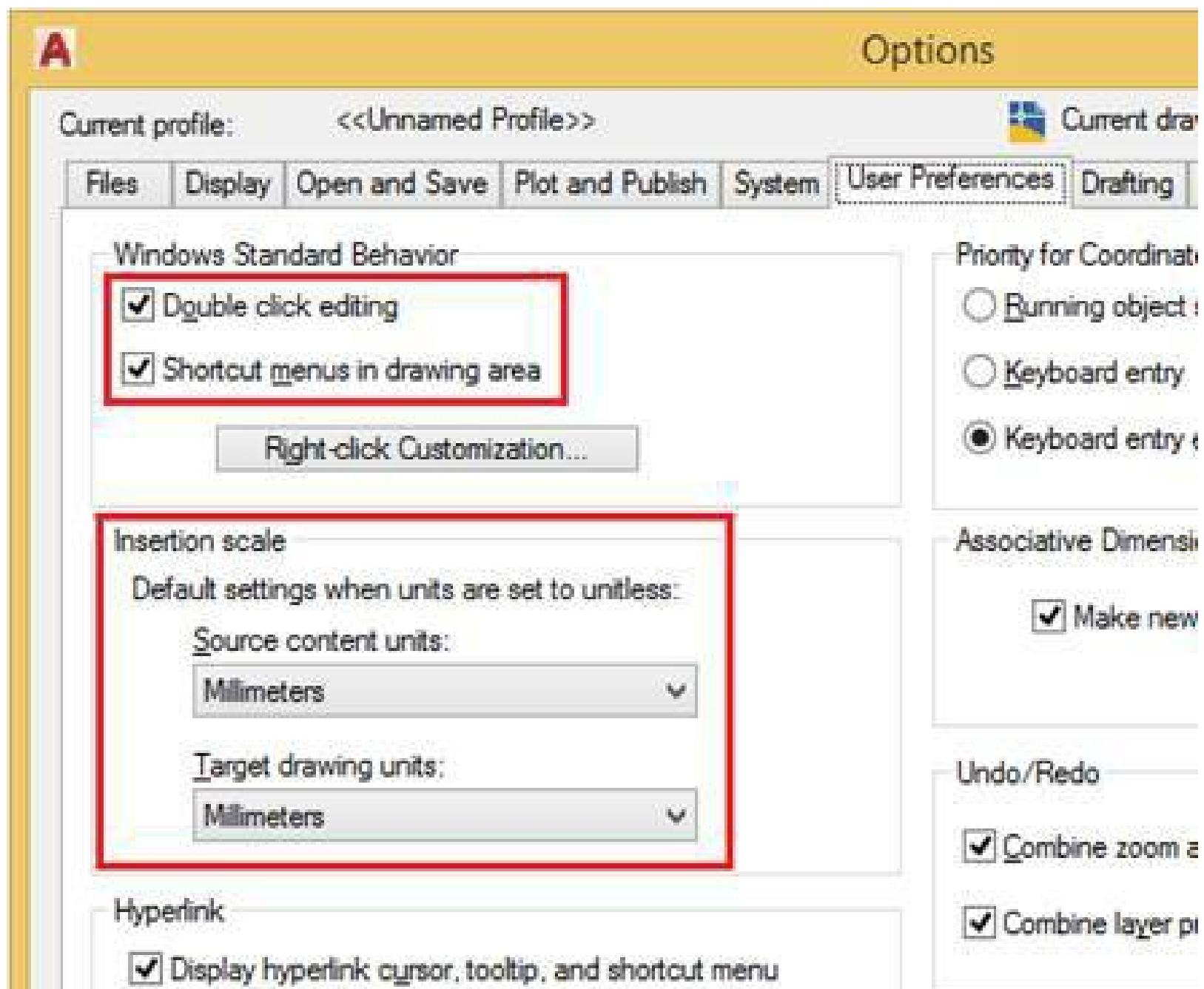


4.3. Plot and Publish

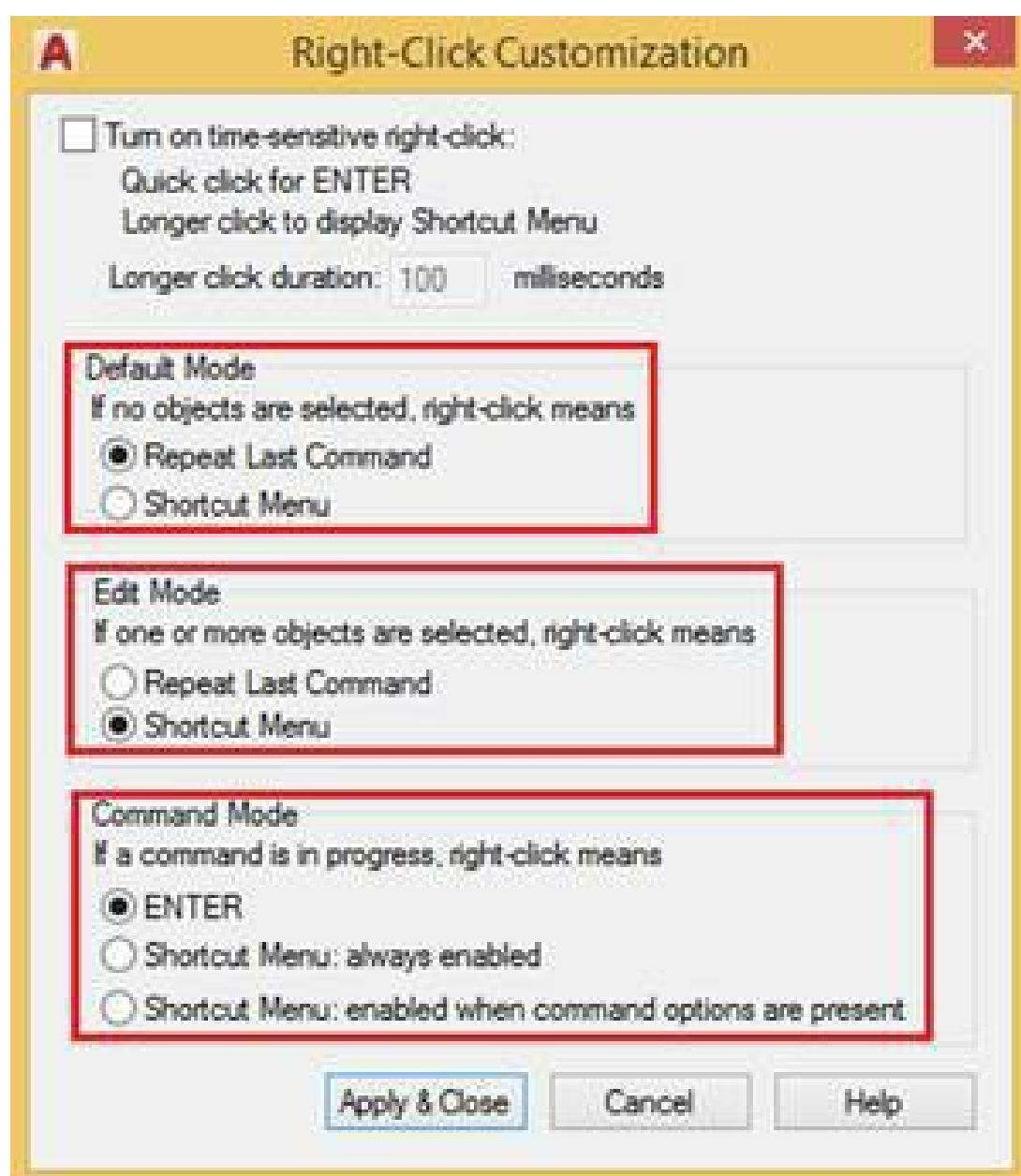


4.4. Use Preference

Setting as follows::



Click on Right-click Customization and install as follows:



4.5. 3D Modelling

A

Options

✕

Current profile: <<Unnamed Profile>>Current drawing: Drawing2.dwg

FilesDisplayOpen and SavePlot and PublishSystemUser PreferencesDrafting3D ModelingSelectionProfiles

3D Crosshairs

☒ Show Z axis in crosshairs

☐ Label axes in standard crosshairs

☐ Show labels for dynamic UCS

Crosshair labels

☒ Use X, Y, Z

☐ Use N, E, S

☐ Use custom labels

X

Y

Z

Display Tools in Viewport

Display the ViewCube

☐ 2D Wireframe visual style

☐ All other visual styles

Display the UCS Icon

☒ 2D Wireframe visual style

☐ All other visual styles

☒ Display the Viewport Controls

3D Objects

Visual Style while creating 3D objects

ByViewport

Deleting control while creating 3D objects

Only delete profile and path curves for solids

Isolines on surfaces

6

 in U

6

 in V

Maximum point cloud points per drawing

10000000

Isellation...

Mesh Primitives...

Surface Analysis...

3D Navigation

☐ Reverse mouse wheel zoom

Walk and Fly ...

ViewCube

Animation ...

SteeringWheels ...

Dynamic Input

☐ Show Z field for pointer input

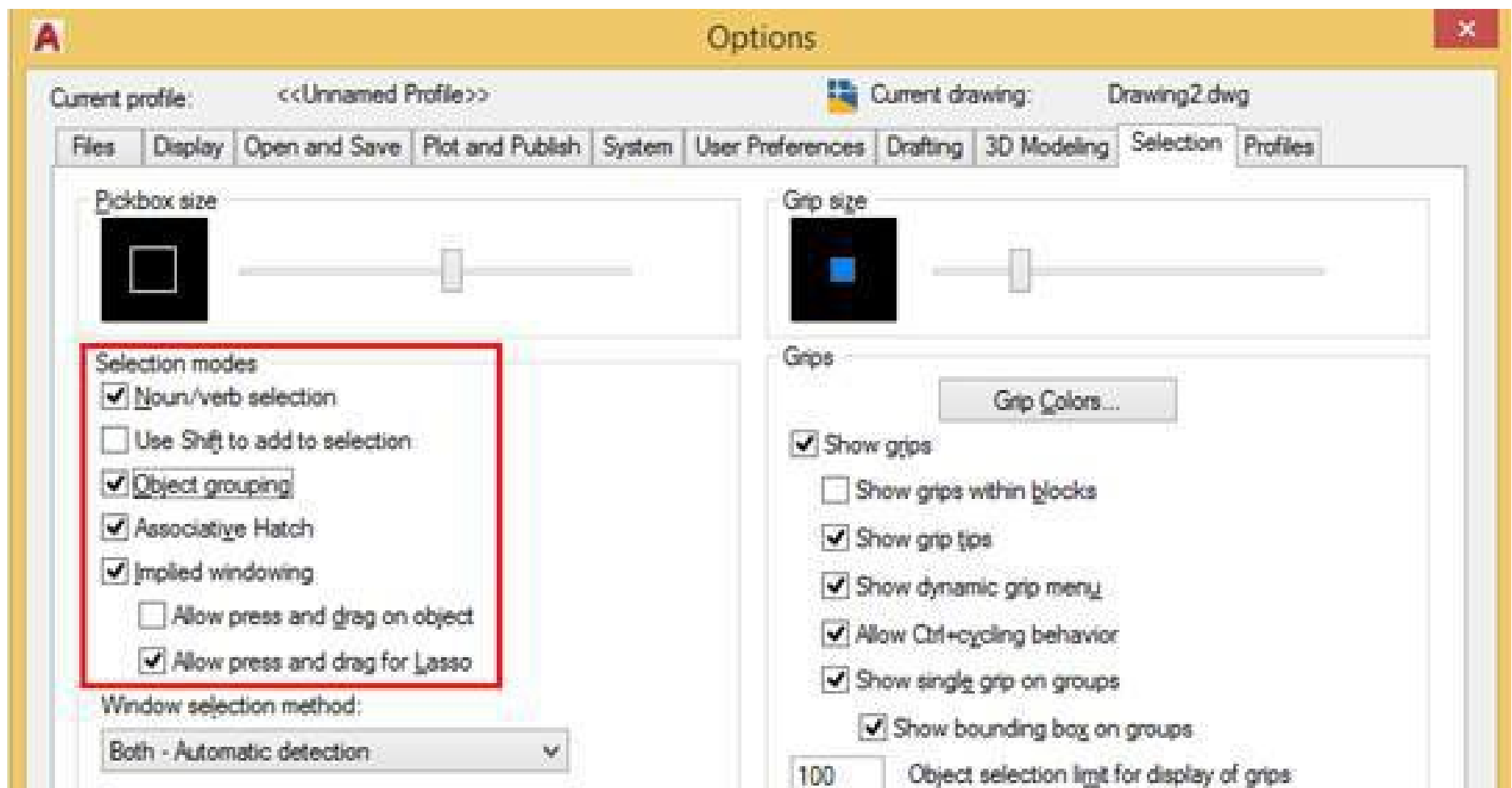
OK

Cancel

Apply

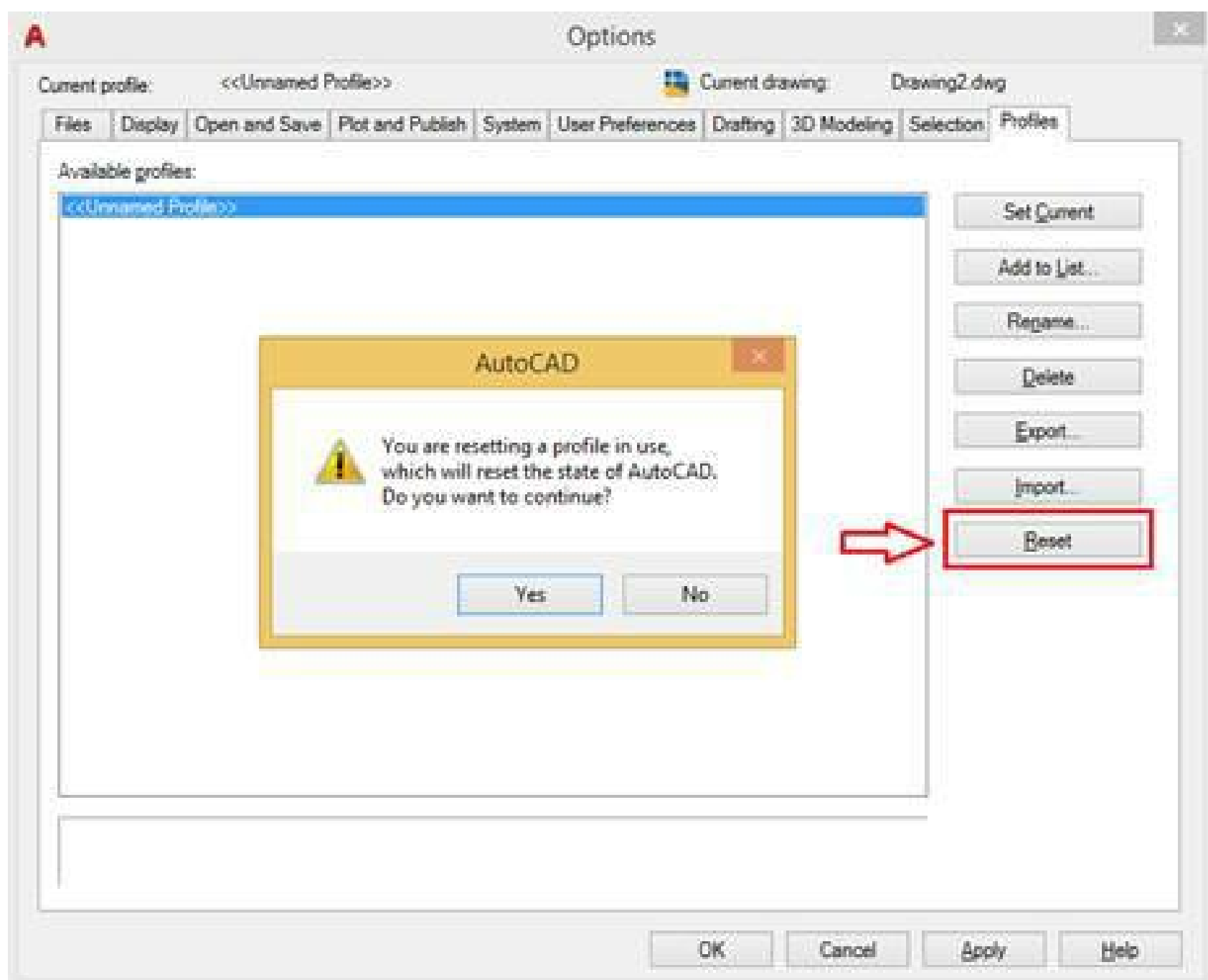
Help

4.6. Selection



4.7. Tab Profile

This tab is not important. There is only 1 item to pay attention to when you need to Reset AutoCAD to default like when you first installed it, then go here. Click Reset, it will bring you back to the beginning. Please consider before pressing the RESET button!



Chapter 5: Professional From Small Things

Some things seem very simple but we don't notice. This leads to a lot of wasted time and unwanted inconvenience.

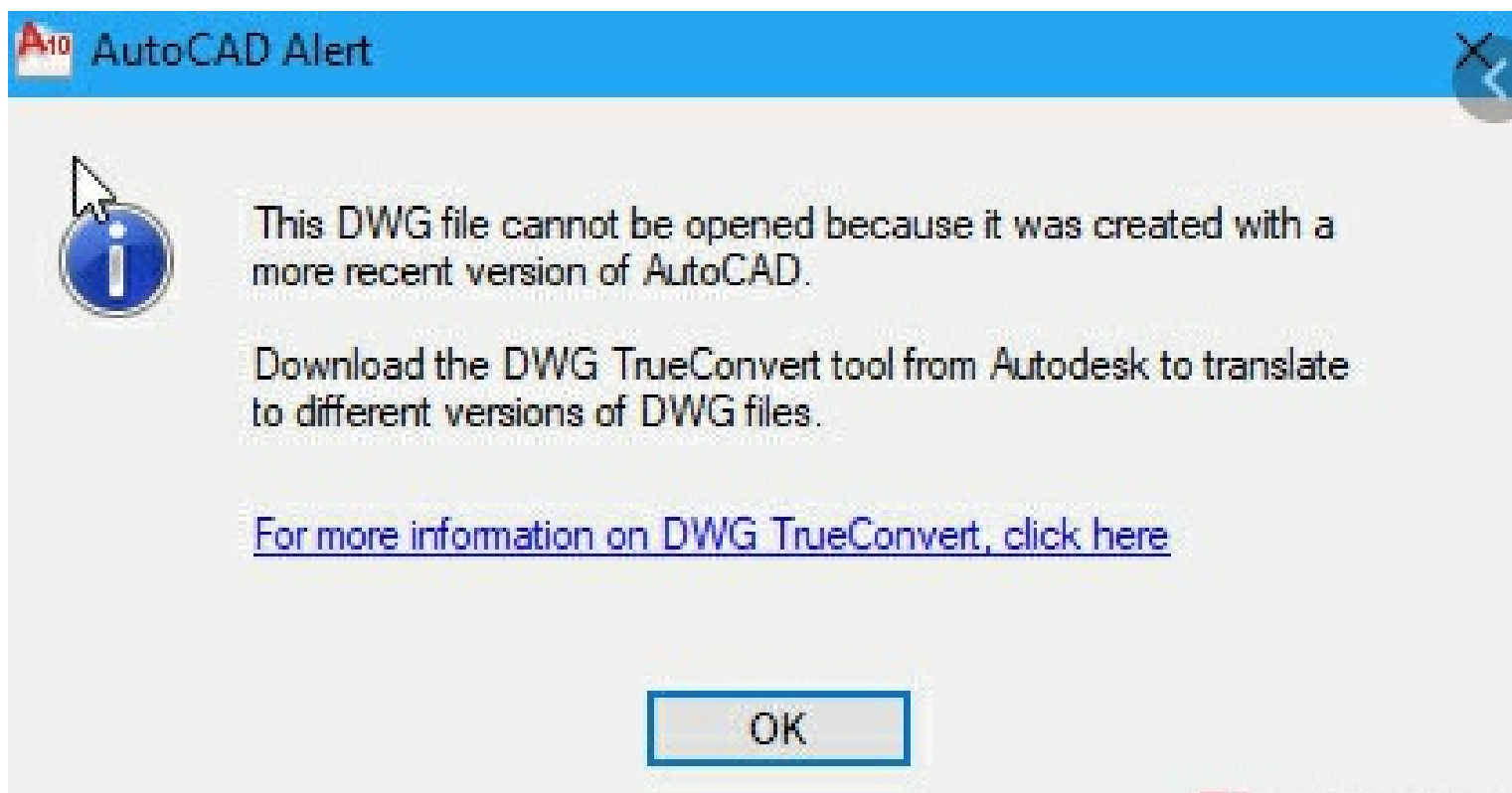
Just like saving drawings.

A lot of people don't know how to save drawings!

And I hate it.

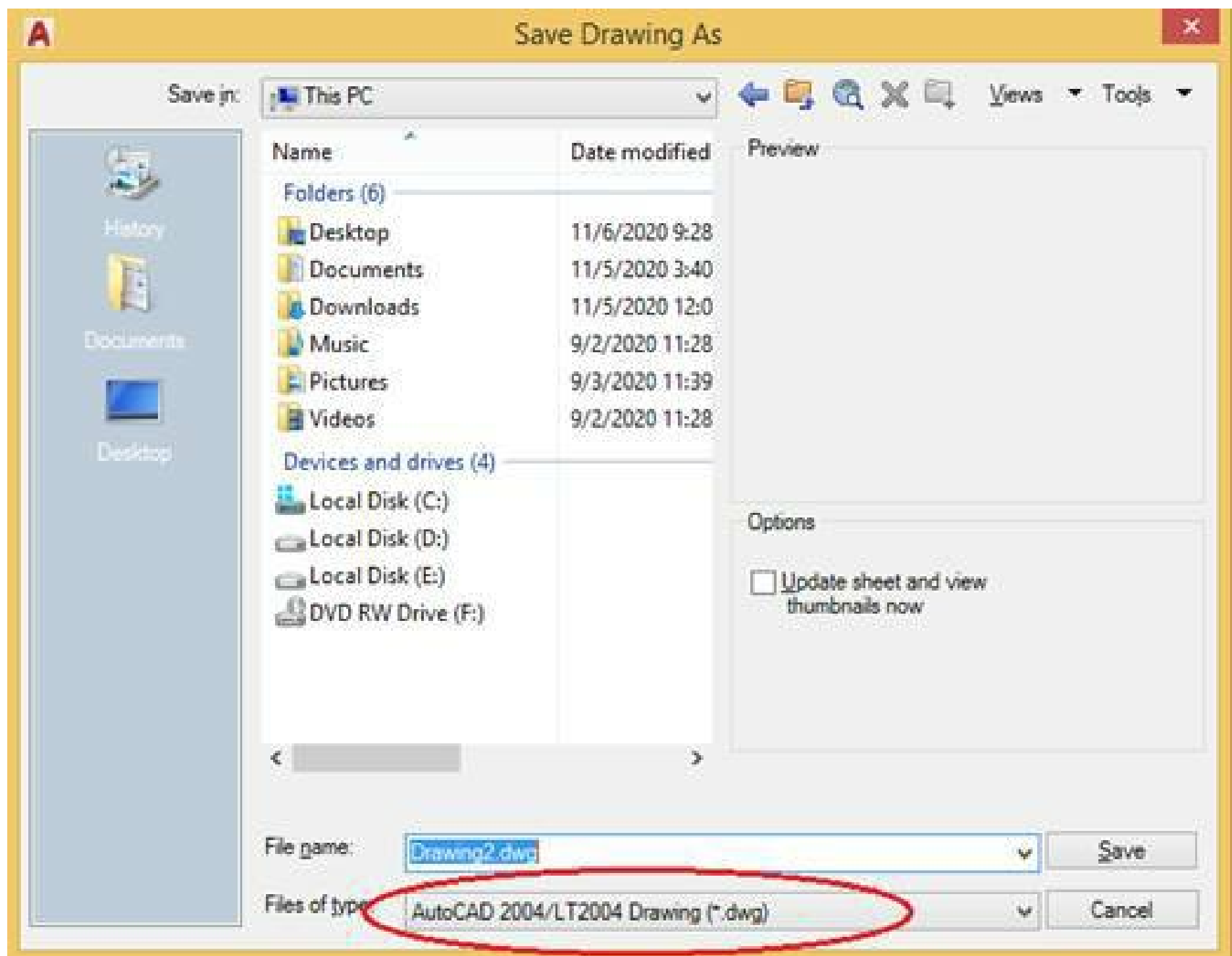
Every one will be as upset as I am if you keep saving the wrong drawing like that!

One thing everyone knows is that AutoCAD can only open drawings in versions equal to or lower than it. Don't know how much CAD the recipient of your drawing is using? And I don't know how many times I have had to receive drawings like this. Click open and it pops up an obnoxious table!



So the minimum rule is to save the version that you think people will read. It only takes 1 click. It's so simple but a lot of people can't do it. Extremely sloppy!

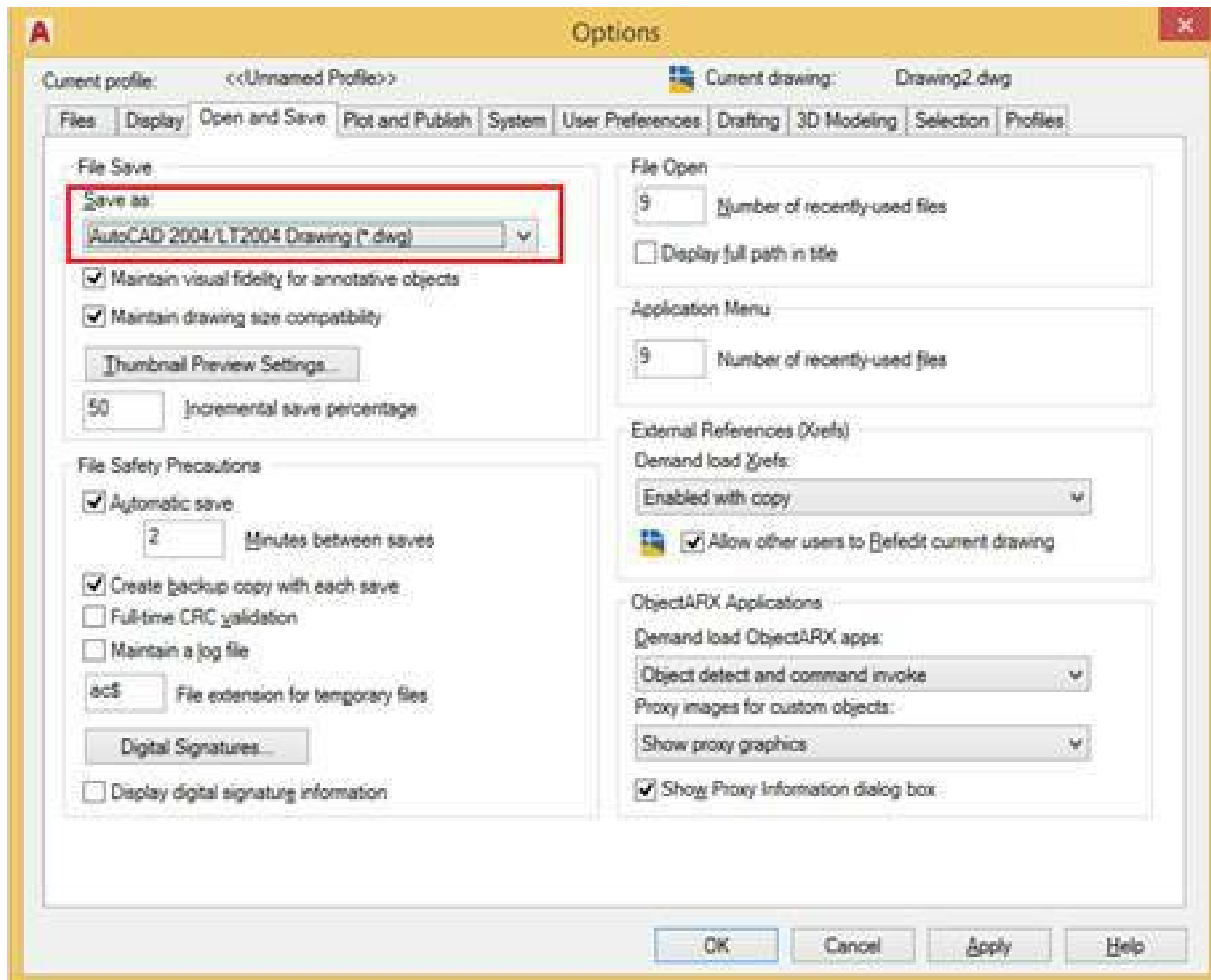
Every time you save the drawing (**Ctrl S**) or save it as a new file (**Shift Ctrl S**), please look at the line Files of type. You click on it, and it is best to choose the saved version of AutoCAD 2004 so that anyone can read it. CAD 2004 users can also read it. Here it is: ***AutoCAD 2004/LT 2004 Drawing (*.dwg)***



One-click is done. Stop sloppy! Be more professional!

But I have a way for you by default to always save about CAD2004.

On the **Open and Save** tab in the Options settings. In the **Save as** section, select the **AutoCAD 2004/LT 2004 Drawing format (*.dwg)**.



After setting here, then next time, when you save the drawing for the first time or save it to another name, it will default to the CAD2004 format, you don't need to click the option again. Do you know what I mean? It's too easy, isn't it?

Please do like that!

Show professionalism from the small things!

Chapter 6: 80/20 Principle

Most Commonly Used Commands

Pareto Principle

It is a fact that each job has its characteristics, and almost only uses a basic set of commands to serve the needs of that job. Here I want to refer to the target group of product design engineers – the work I have done before, what basic commands will they often use?

In the book “AutoCAD Basics for Beginners”, there is a very good sentence that “80% of real-world situations are solved by only 20% of the features, the key point is knowing what 20% of the features are.” which command”.

The Pareto Law comes from Vilfredo Pareto, an Italian economist who believes that 80% of Italy's wealth and land is held by 20% of the population. This law is gradually extended to other fields. For example, 80% of costs come from 20% of resources, 80% of profits come from 20% of customers, or 80% of happiness comes from 20% of people around.

The Pareto principle with AutoCAD also shows that you don't need to know too many commands, just knowing 20% are the most practical commands, you have already handled 80% of your work.

AutoCAD mechanical does not need to use too many commands. With only about 20 commands that you need to use proficiently.

- Common commands: LINE, MOVE, COPY, SCALE, TRIM, DIMLINEAR, DIMRADIUS, DIMDIAMETER, MATCHPROP, MTEXT, REGEN, ERASE.
- Less common commands: EXTEND, BLOCK, EXPLODE, MLEADER, OVERKILL, EXTRIM, PURGE, ...

In general, the statements are just simple, not complicated.

And this book is of course not intended to teach you how to use commands.

Chapter 7: Change and Add Command for AutoCAD

I'm about to reveal to you a secret!

I have become interested and passionate about AutoCAD since learning this secret.

If your work often uses AutoCAD, then this is a superior weapon.

It helps you to work with software lightning fast!

Viewers will just roll their eyes because your manipulation is like a "sparrow".

You will dance and glide on the keyboard like you are immersed in a ballet of your own.

Most importantly, it helps you increase your productivity by 3 times.

Save time and increase work inspiration

Simple and very easy to do.

I bet it is!

Do you want to know what that secret is?

Let's add commands and combine them with change commands!

Example 1:

A line has to command L. But since the L key is too far away, I add the command Z = L = Line. Then, pressing Z or pressing L is both a Line command request.

The problem arises: Then the Z command will no longer call the Zoom command. However, in terms of priority, the Line command is used more than the Zoom command because I mainly zoom with the mouse. I will still add a shortcut to call the Zoom command A.

Thus: The Line command has added a shortcut of Z. And the Zoom command is changed to a shortcut from Z -> A.

Example 2:

The Copy command has a shortcut command of CO or CP, while I have to use this command a lot. Typing CO/CP takes time because the O and P are far away, I add the command Q = CO = CP = Copy.

When needed, I type Q to copy. When others use it, they type CO/CP to copy. Who do you think is faster?

I have used this method for many years and can confirm, adding commands and changing commands “appropriately” is one of the few most important techniques for fast AutoCAD drawing or fastest AutoCAD drawing presentation.

Each of your commands will be performed as fast as a rattlesnake's bite.

And from this moment on, that secret is officially yours!

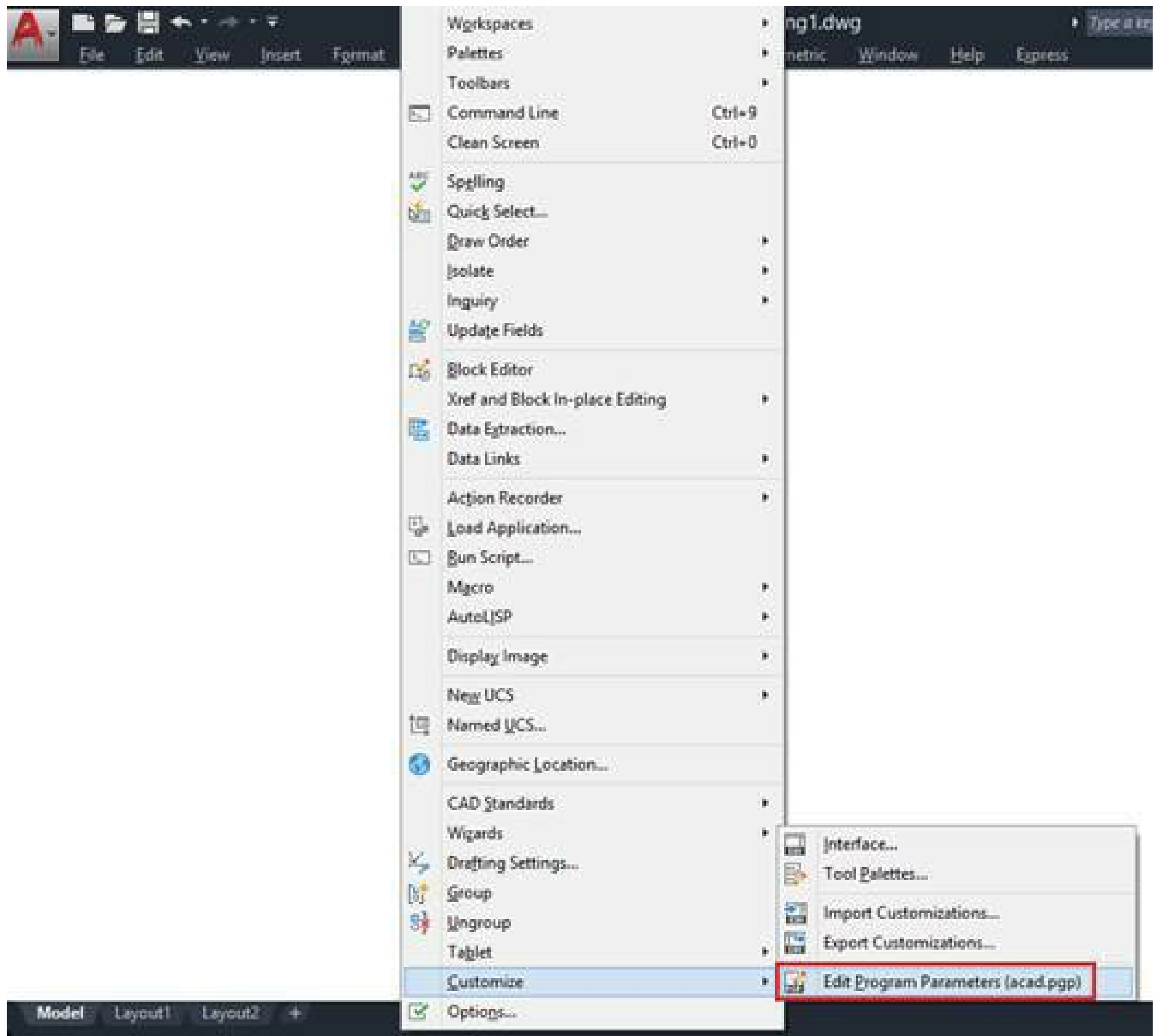
The principle of adding commands and changing commands is simple: “Always put the most used commands to the left-hand side of the keyboard and memorize them”. That's it!

You can change so that only you can use your computer. However, I do not recommend this, so limit, instead of changing the command, add a command so that someone can still use your machine. Unless required.

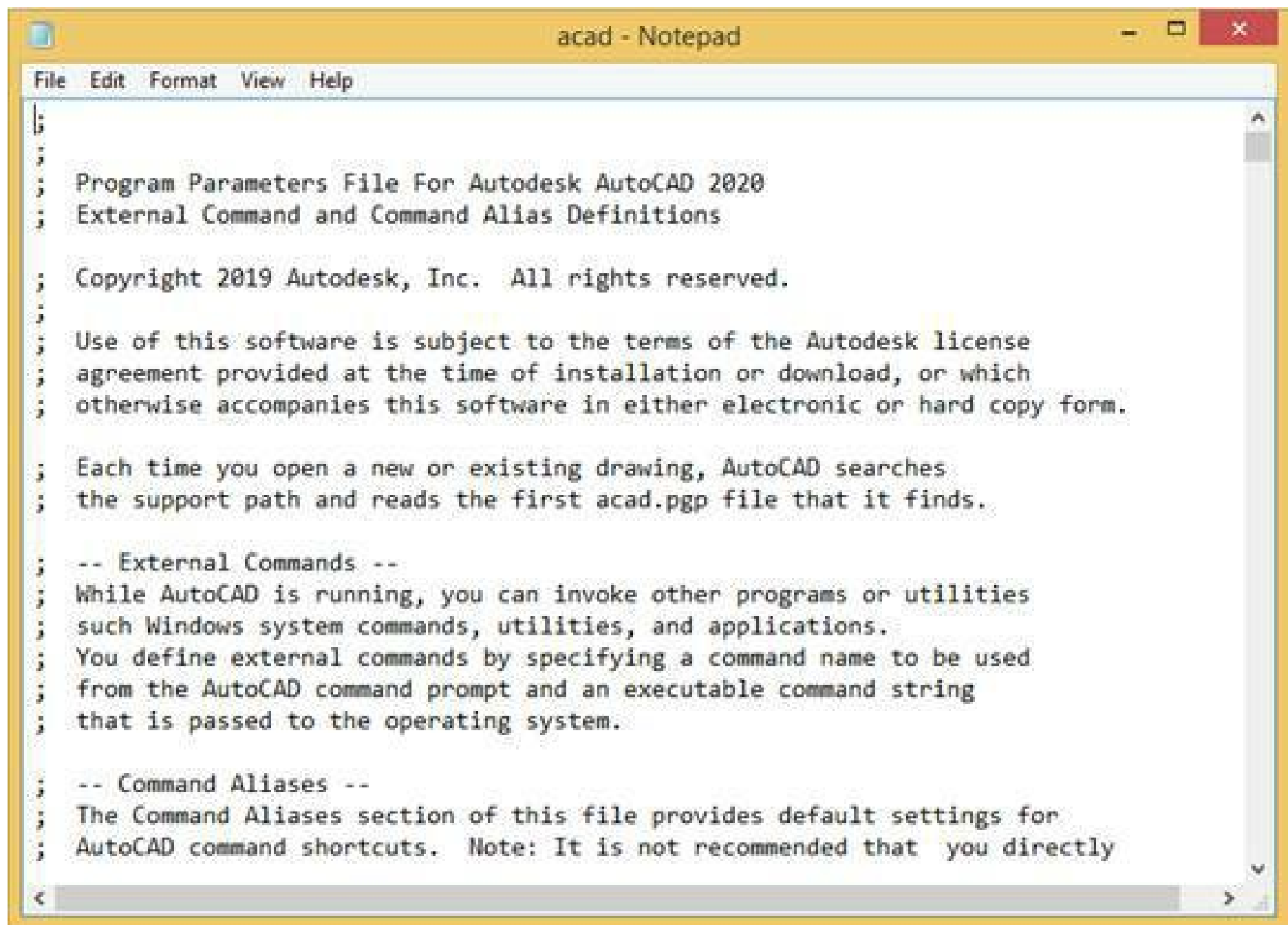
So how to edit the AutoCAD command, I would like to introduce the simplest way as follows. There are other ways but this is the fastest and easiest to manage.

- *Tools/ Customize/ Edit Program Parameters* (acad.pgp)

(or typing key combination: **Alt T C P P**)



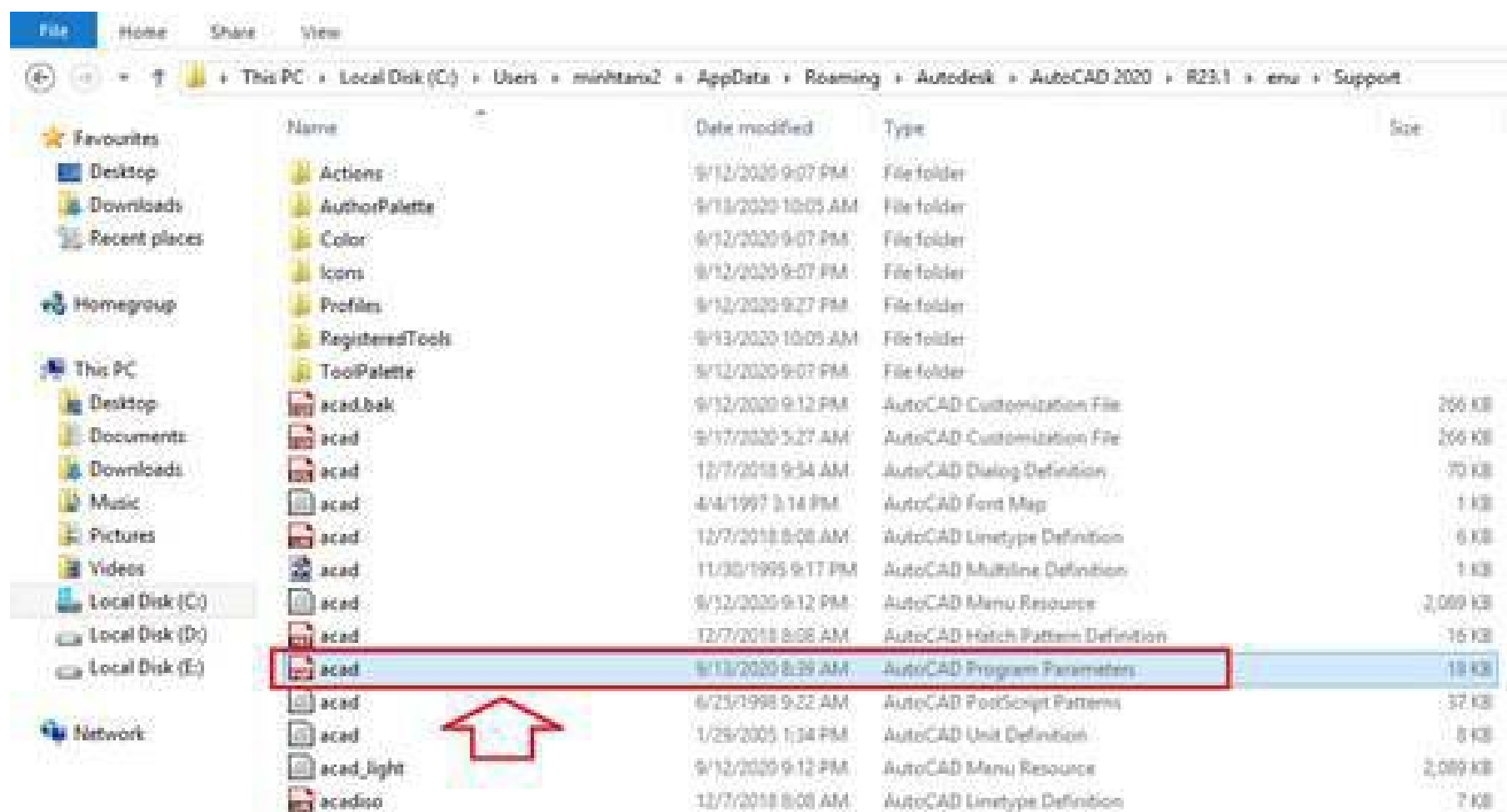
AutoCAD will open the file acad.pgp



```
File Edit Format View Help
;
;
; Program Parameters File For Autodesk AutoCAD 2020
; External Command and Command Alias Definitions
;
; Copyright 2019 Autodesk, Inc. All rights reserved.
;
; Use of this software is subject to the terms of the Autodesk license
; agreement provided at the time of installation or download, or which
; otherwise accompanies this software in either electronic or hard copy form.
;
; Each time you open a new or existing drawing, AutoCAD searches
; the support path and reads the first acad.pgp file that it finds.
;
; -- External Commands --
; While AutoCAD is running, you can invoke other programs or utilities
; such Windows system commands, utilities, and applications.
; You define external commands by specifying a command name to be used
; from the AutoCAD command prompt and an executable command string
; that is passed to the operating system.
;
; -- Command Aliases --
; The Command Aliases section of this file provides default settings for
; AutoCAD command shortcuts. Note: It is not recommended that you directly
```

Where is this file located?

With AutoCAD 2020 on my computer, the path is:



(You must choose to show hidden files to see the AppData folder)

***Command change syntax::**

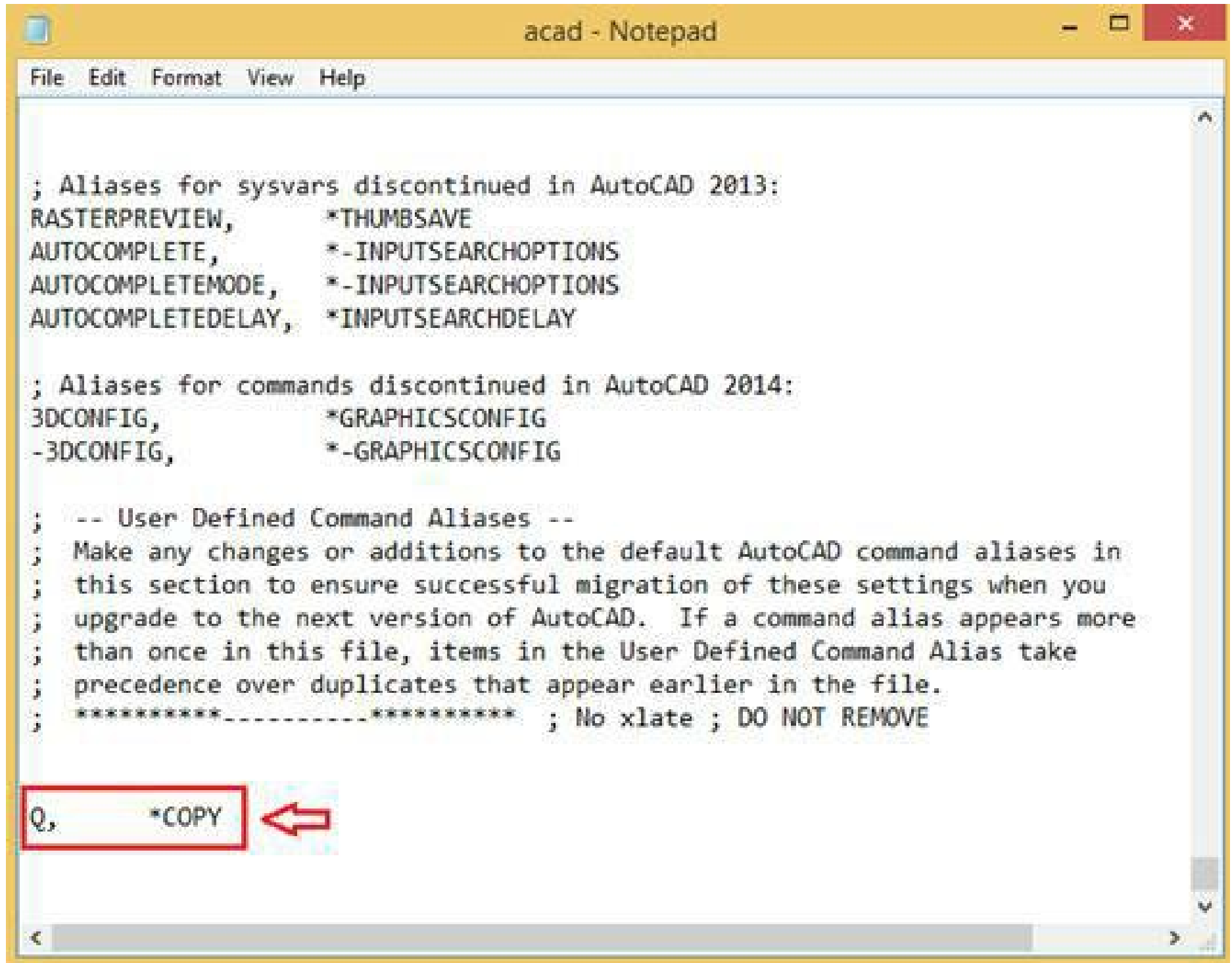
- Move the cursor to the bottom empty area of the file acad.pgp

- Enter the content to change the command as:

< **New shortcut command** > <comma> <Tab> <Star> <full name of the command you want to change>

For example, I want the Q command to add to the COPY command (its old command is: CO, CP, COPY), I enter the content for the acad.pgp file:

Q, *COPY



```
; Aliases for sysvars discontinued in AutoCAD 2013:
RASTERPREVIEW,      *THUMBSAVE
AUTOCOMplete,        *-INPUTSEARCHOPTIONS
AUTOCOMpleteMODE,    *-INPUTSEARCHOPTIONS
AUTOCOMpleteDELAY,   *INPUTSEARCHDELAY

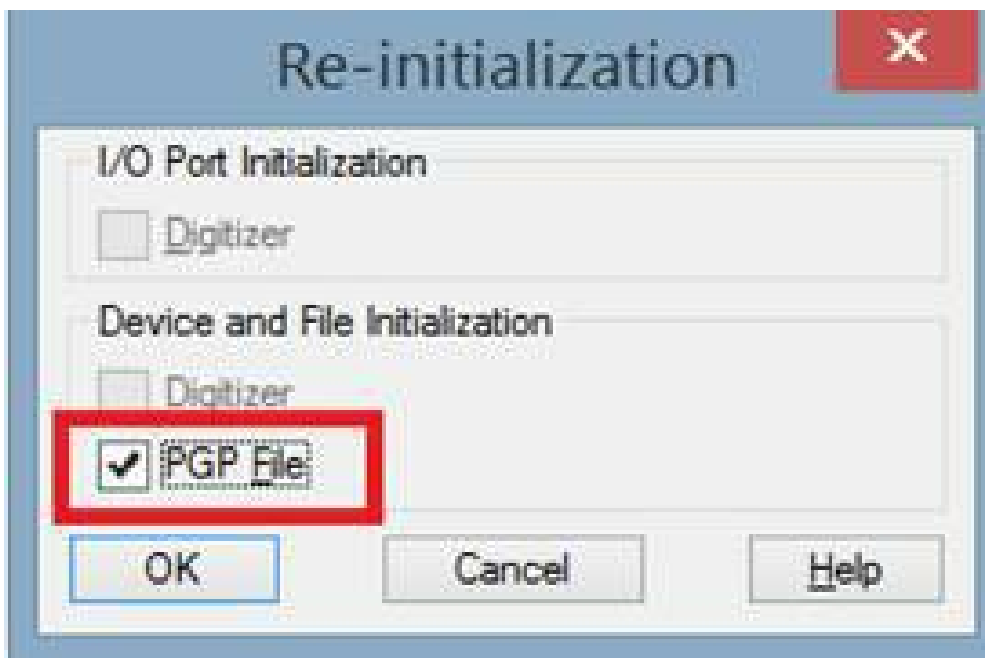
; Aliases for commands discontinued in AutoCAD 2014:
3DCONFIG,            *GRAPHICSCONFIG
-3DCONFIG,           *-GRAPHICSCONFIG

; -- User Defined Command Aliases --
; Make any changes or additions to the default AutoCAD command aliases in
; this section to ensure successful migration of these settings when you
; upgrade to the next version of AutoCAD. If a command alias appears more
; than once in this file, items in the User Defined Command Alias take
; precedence over duplicates that appear earlier in the file.
; *****-----***** ; No xlate ; DO NOT REMOVE

Q, *COPY
```

Note: If the added command coincides with another available command of AutoCAD, then you change the duplicate command to another command to replace it.

- After changing the command / select **Close** to save the acad.pgp file / Select **Save** to save
- Return to AutoCAD, type the command **REINIT**✓
- Check the box before the line "PGP File"
- OK for the change to take effect.



Now we have successfully changed the order. Follow along to verify for yourself. Practice now to boost your learning performance!

Here is my add and change table, which is still very long at the bottom.

```

; -- User Defined Command Aliases --
; Make any changes or additions to the default AutoCAD command aliases in
; this section to ensure successful migration of these settings when you
; upgrade to the next version of AutoCAD. If a command alias appears more
; than once in this file, items in the User Defined Command Alias take
; precedence over duplicates that appear earlier in the file.
; ***** ; No xlate ; DO NOT REMOVE
;
0,      *DIMCENTER
00,     *DIMCEN
1,      *LAYON
2,      *MOVE
1',     *LAYOFF
3,      *DIMLINEAR
4,      *LAYMCUR
5,      *STYLE
54,     *LAYCUR
6,      *REINIT
7,      *LAYISO
8,      *LAYTHW
9,      *DIMORDINATE
98,     *LAYULK

--,     *PLOT
11,     *OPEN
22,     *DIMRADIUS

```

Adding-changing this command is according to the personal preferences and the specific work of each person, so I do not force you to follow a set of shortcuts at all. As long as you feel most comfortable, don't be forced by anyone, just be yourself. But do not forget the original command, in case we use another computer.

Chapter 8: Succinct Layer Management

1. The need to manage objects by Layer

Layer management is the arrangement of objects with common characteristics and properties into a group, called a Layer. Each layer will carry its properties, including layer name, color, line style, density, ...

Unlike construction drawings that have many types of objects with many different properties, there are drawings with several dozen layers that are normal, layers are also named according to rules for easy searching. In my mechanical drawing I found that there are quite a few groups of objects, so layer management is simple.

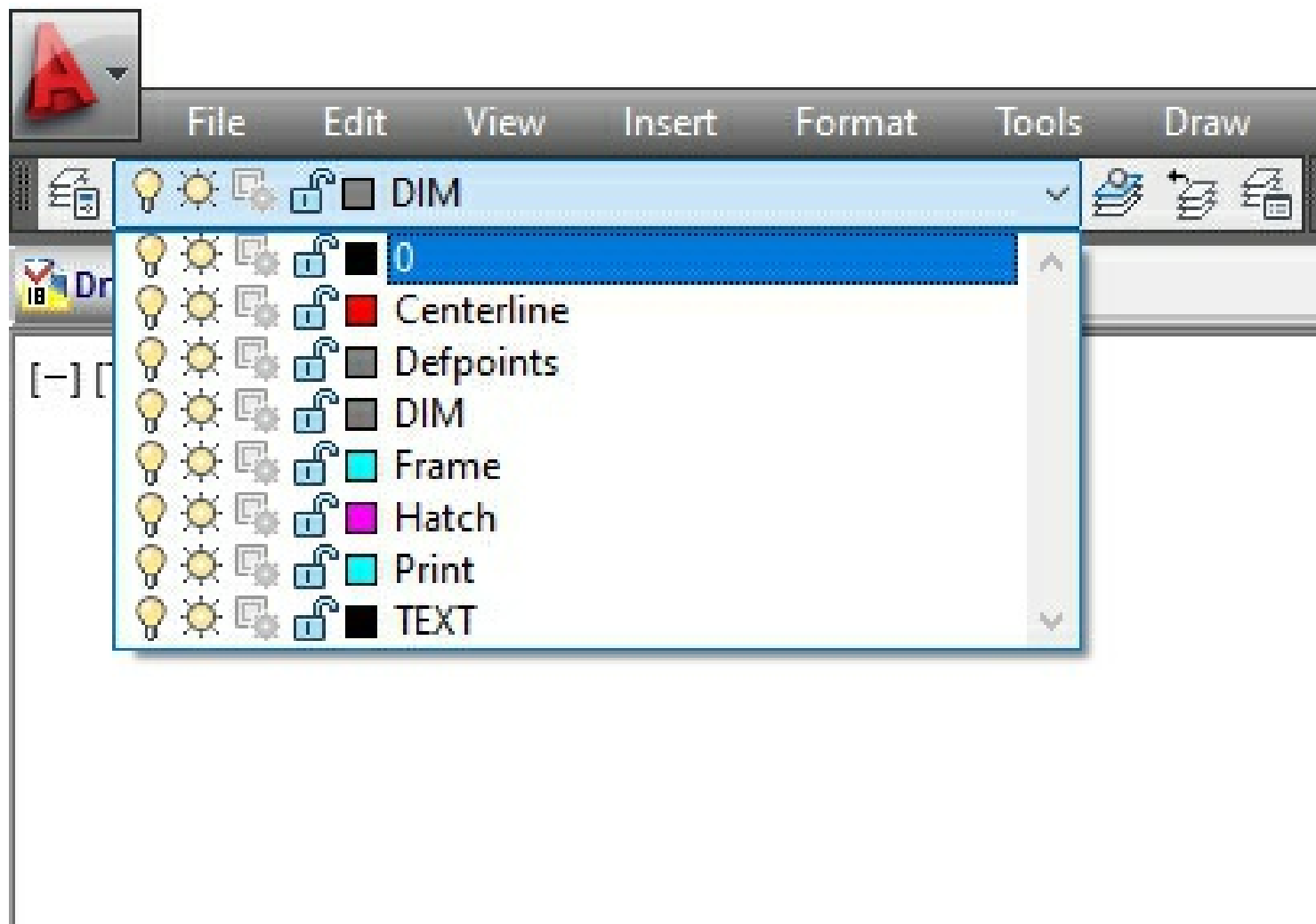
However, managing objects by layer are still necessary. Good layer management, you select objects easily. You plan the object, manage the object. Every small action on the drawing shows your professionalism and meticulousness. And you will be well rewarded!

Num	Linename	Linetype	Color	Lineweight (mm)				
				A4	A3	A2	A1	A0
1	Visible line		 White	0.3	0.35	0.4	0.6	0.8
2	Hidden line		 Grey	0.15	0.2	0.25	0.3	0.4
3	Center line		 Red	0.15	0.2	0.25	0.3	0.4
4	Reference line		 Grey	0.15	0.2	0.25	0.3	0.4
5	Dimension line		 Grey	0.15	0.2	0.25	0.3	0.4
6	Extension line		 Grey	0.15	0.2	0.25	0.3	0.4
7	Sectional brick		 Purple	0.15	0.2	0.25	0.3	0.4
8	Table, Title block		 Cyan	0.3	0.35	0.4	0.6	0.8
9	Detail		 Grey	0.15	0.2	0.25	0.3	0.4

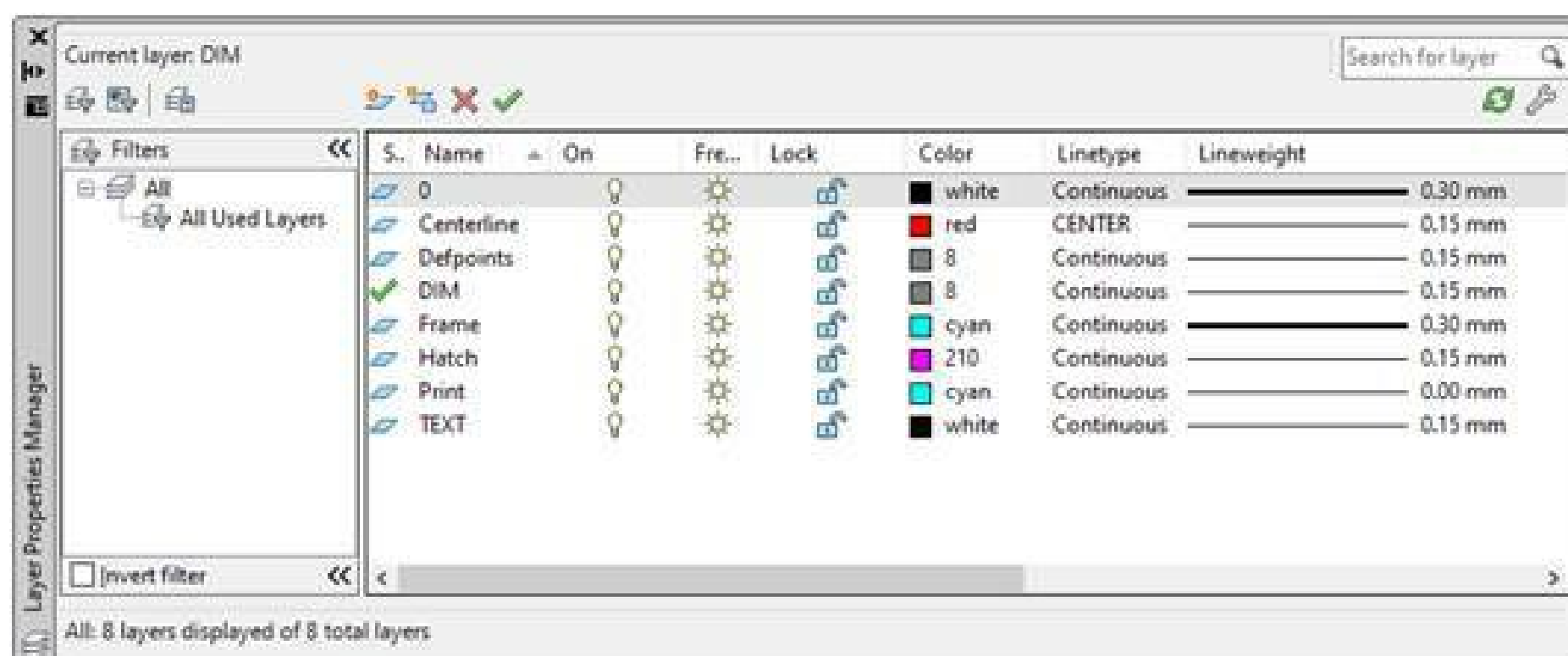
Refer to the Table of regulations on the lines used on the drawing

For my work, the way that I am applying, I usually classify and arrange into main layers as follows:

- ü Group of objects as projection:
 - ü Group of objects as dimension lines:
 - ü Group of objects as hatch strokes:
 - ü Group of notes and technical requirements:
 - ü Grouping objects to indicate the mass printing area:
 - ü Group of objects visible on the drawing but not printed:
 - ü Group of objects located on the title block:
 - ü Group objects as the centerline cut line, the axis of symmetry, the contour of detail:
- layer 0
 - layer DIM
 - layer HATCH
 - layer TEXT
 - layer PRINT
 - layer Defpoints
 - layer FRAME
 - layer CENTERLINE



An example of my layer management panel (lineweight for A4 size)



Num	Layer	Color		Linetype	Lineweight (mm)				
		Name	Code		A4	A3	A2	A1	A0
1	0	White	07	Continuous	0.3	0.35	0.4	0.6	0.8
2	Centerline	Red	01	Center	0.15	0.2	0.25	0.3	0.4
3	Defpoints	Gray	08	Continuous	0.15	0.2	0.25	0.3	0.4
4	Dim	Gray	08	Continuous	0.15	0.2	0.25	0.3	0.4
5	Hatch	Magenta	210	Continuous	0.09	0.15	0.15	0.2	0.25
6	Frame	Cyan	04	Continuous	0.3	0.35	0.4	0.6	0.8
7	Text	White	07	Continuous	0.15	0.2	0.25	0.3	0.4
8	Print	Yellow	02	Continuous	0	0	0	0	0

To access the layer management panel, type the command: LA ✓



In this panel, you can create new, edit properties, or delete layers. After creating any new layer, you need to fully set the properties of that layer. I will not repeat it because this is basic knowledge.

On a drawing, how to convert objects with the same properties to a layer?

- Move left mouse and scan to select the objects you need to convert
- Click on the Layers toolbar, in the drop-down list of layers, click on the name of the layer you want to move the objects to. That's OK!

Assignment: (required to complete)

Make a layer management board according to the standards of the company/organization you are working for and put it in your Template file.

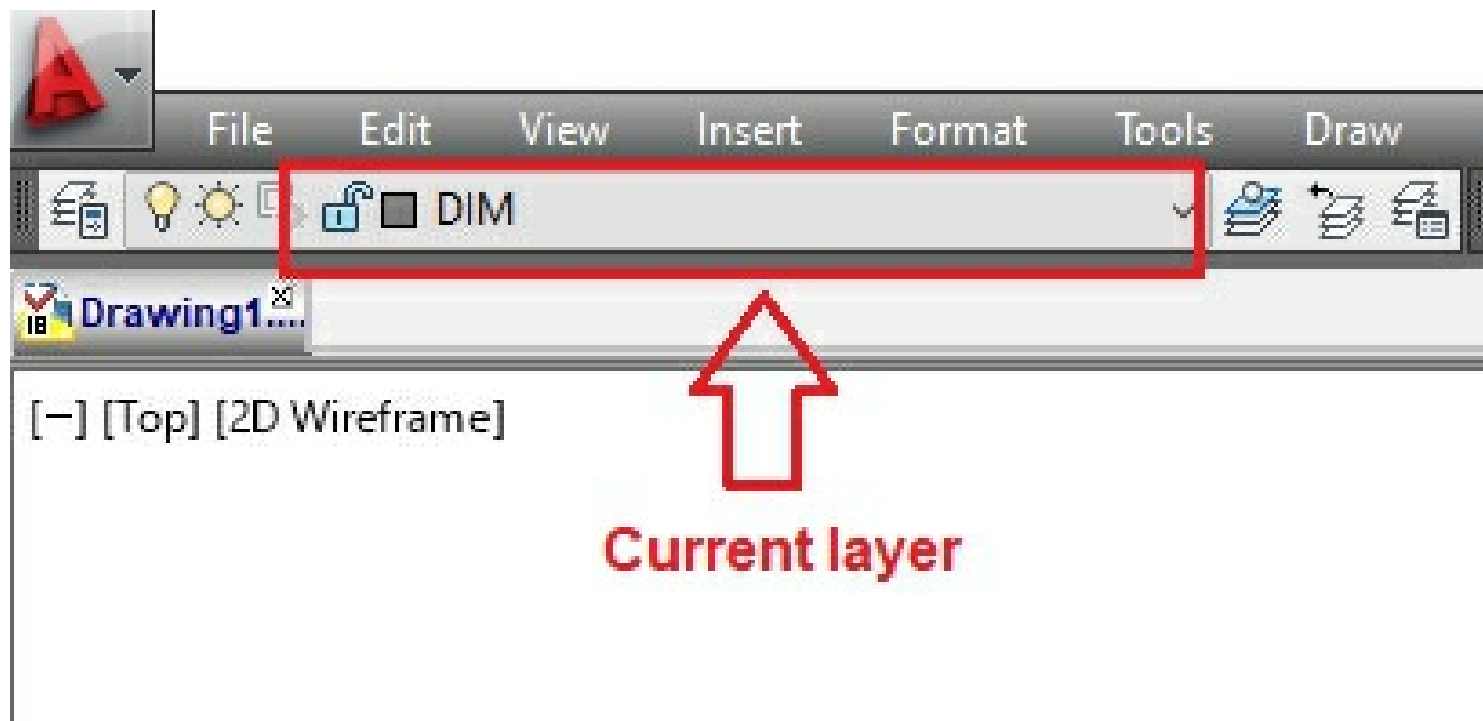
You can complete your exercise right here.

2. Current Layer

Once you have managed layers, you need to get used to the concept of "current layer".

The active layer is the layer that is standing on the drawing board. When you draw any new object, that object will belong to the current layer. You can optionally select any layer as the active layer.

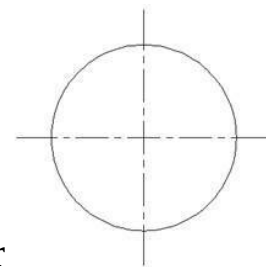
How to detect the active layer: The active layer is permanently visible in the layers toolbar



To select the current layer

Method 1: Click on the layers toolbar and select the name of the layer you want to make the current layer.

Method 2: Type LA↵ to access the Layer management panel, double-click on the layer name you want to make active (a blue checkmark appears at the top).



Example: Ask to draw any circle as follows and bring them to the correct layer

Here there are 3 objects, divided into 2 categories:

- Circle, layer 0
- 2 centerline objects, Centerline layer

The steps are as follows:

- On a new drawing file, create two new layers with properties according to the convention table.
- Make layer 0 the active layer
- Draw a circle
- Make the Centerline layer the active layer
- Draw 2 center lines.

It is done!

Also, you can initially draw all 3 objects on layer 0. Then select 2 centerline objects and convert them to the Centerline layer.

3. The most commonly used layer management commands

The group of command tools for layer management alone has dozens of commands. However, within the scope of my job as a mechanical engineer, I have filtered out just the 3 most commonly used commands that you need to memorize and use every day.

The 3 most popular layer commands are LAYOFF, LAYON, and LAYMCUR.

Other commands you may not know because they are not important or used very little.

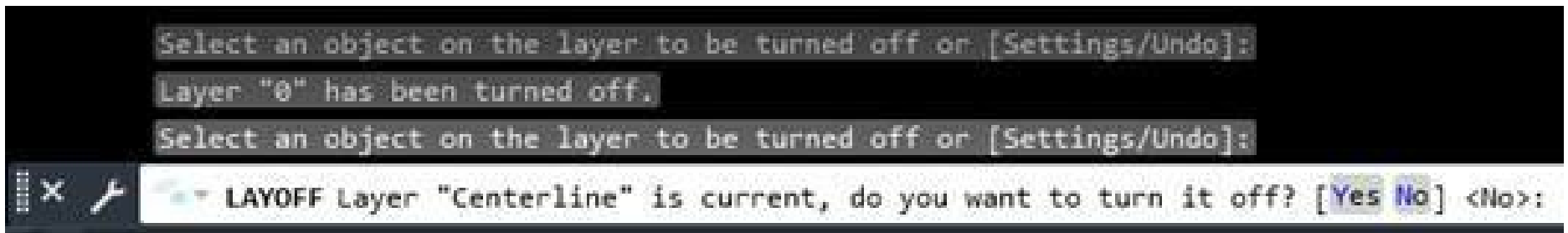
3.1. LAYOFF - Hide one or more selected layers

- When you have classified objects by layer, each object on the drawing will belong to a certain layer managed by you. When you want to hide a layer ie hide all the objects belonging to that layer, you use the LAYOFF command.

Application of LAYOFF: Hide 1 or more selected layers

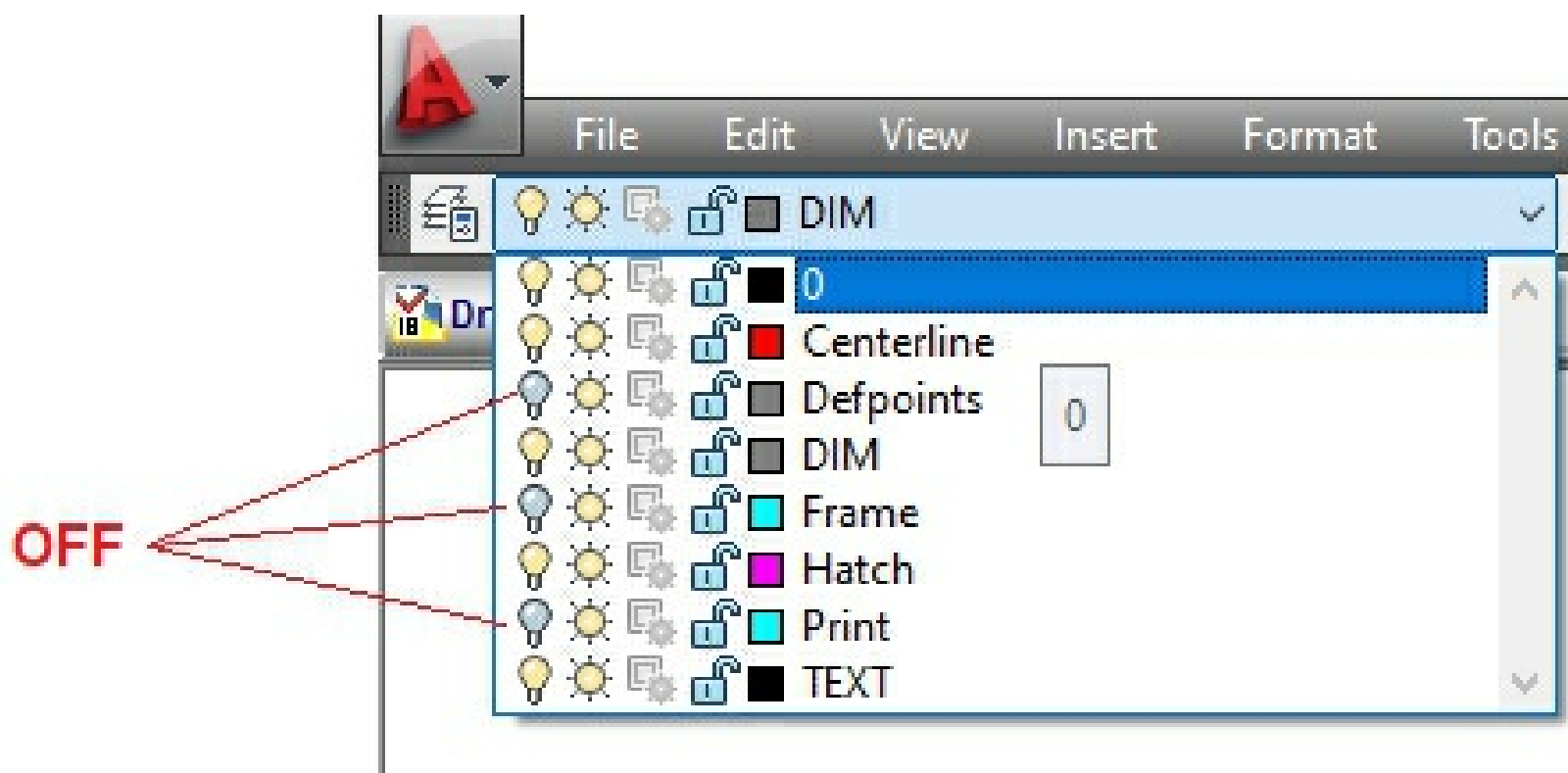
How to use:

- Type **LAYOFF** ✓
- At the prompt “*Select an object on the layer to be turned off:*” you click on any object on the layer you want to hide.
- If the object belongs to the current layer, AutoCAD will ask you to confirm if you want to hide the current layer.



How to know the layer is turned off (Layoff)?

- In the Layers toolbar, if the light in front of a layer is turned off, that layer is disabled (Layoff).



- The sign is similar in the Layers management panel, but the light bulb position is moved to the back.

The shortcut command of LAYOFF that I set myself is 1`. Extremely favorable position on the keyboard, you know enough about the importance of this LAYOFF command. Try it!

3.2. LAYON – Turn on all layers on the drawing

- Function: Turn on all layers in the drawing, except for the frozen layer (Freeze). Then, the layers that are turned off (LAYOFF) will all show up (from OFF to ON).

So:

LAYON is the opposite of LAYOFF. LAYON is on all (except Freeze), LAYOFF is off selectively.

3.3. LAYMCUR – Set current layer

You can select the current layer by selecting it on the Layers toolbar. In the case of a drawing with several dozen layers, finding one of them is sometimes not easy. Therefore, LAYMCUR was born to solve the above problem.

LAYMCUR has the effect of setting a layer as the current layer by selecting any object on that layer.

How to use the command:

- **LAYMCUR** ✓
- At the prompt “*Select object whose layer will become current*”: You select any object on the layer you want to make the current layer.

OK! I have introduced to you the 3 most commonly used Layer management commands.

In addition, I would like to introduce to you one more keyword, which is the **LAYTRANS** command.

That's great too!

Chapter 9: Insert Note Content and Manage Text style

1. Notes using Mtext and Text

1.1. Should we use Mtext or Text?

On any drawing, the following contents are indispensable: Technical requirements, instructions on content, information in the drawing frame, ...

There are two styles of writing on AutoCAD that are written in MTEXT or TEXT.

	MTEXT	TEXT
Command	T	DT
Height Management	Double click and edit in the Text Formatting bar	Edit the height in the Properties dialog box
Change color		Edit the color in the Properties toolbar
Insert special characters or special formatting		Cannot
Notes created into a block		Cannot
Capacity increase of the drawing	With the same amount of content, a drawing containing Mtext will be heavier than a drawing using only Text	

Recommendation: MTEXT has more advantages. We should use Mtext to write in AutoCAD drawings because it is easy to edit and format as you like.

1.2. How to change Text to Mtext and vice versa

- Change Text to Mtext: you use the command TXT2MTXT
- Change MText to Text: you use the Explode command (X)

2. Text style Manager

Text style is the type of font you write on the drawing, including the management of 3 main parameters:

- Font
- Text hight
- Width factor

Num	Content	Font	Color	Text height (mm)				
				A4	A3	A2	A1	A0
1	Technical requirements	Arial	White	2.5	3.5	5	7	10
2	Notes, information	Arial	White	2.5	3.5	5	7	10
3	Digits used to note the dimension	Iso, Block font	Purple	2.5	3.5	5	7	10
4	Digits used to note the ordinal number of the part	Iso, Block font	Red	2.5	3.5	5	7	10

Refer to the table of regulations on the height of letters according to the title block

You just need to create and manage the following 2 Textstyles for 1 paper size:

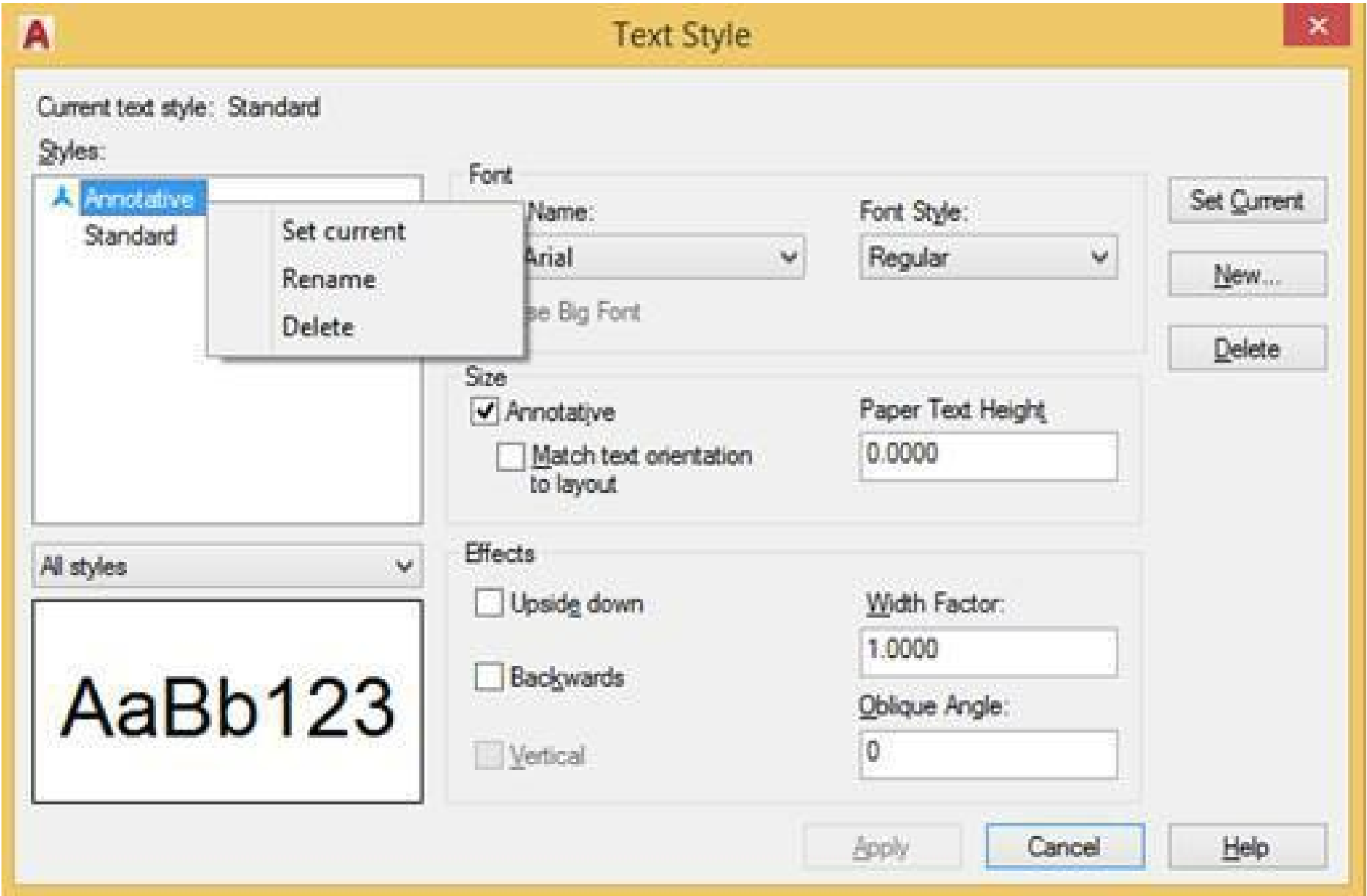
Text style	Apply for	Font Name	Font Style	Height (mm)		Width Factor
				A4	A3	
	- Write the content in the title block					

Textnote	- Write all other notes on the drawings (Technical requirements, section names, excerpts, notes, change history, lists, ...)	Arial	Regular	2.5	3.5	1
Textdim	- Dimension - Numbering when raising Revision - Numbering the parts of the assembly drawings	iso.shx	Regular	2.5	3.5	0.8

2.1. Create a new Text style

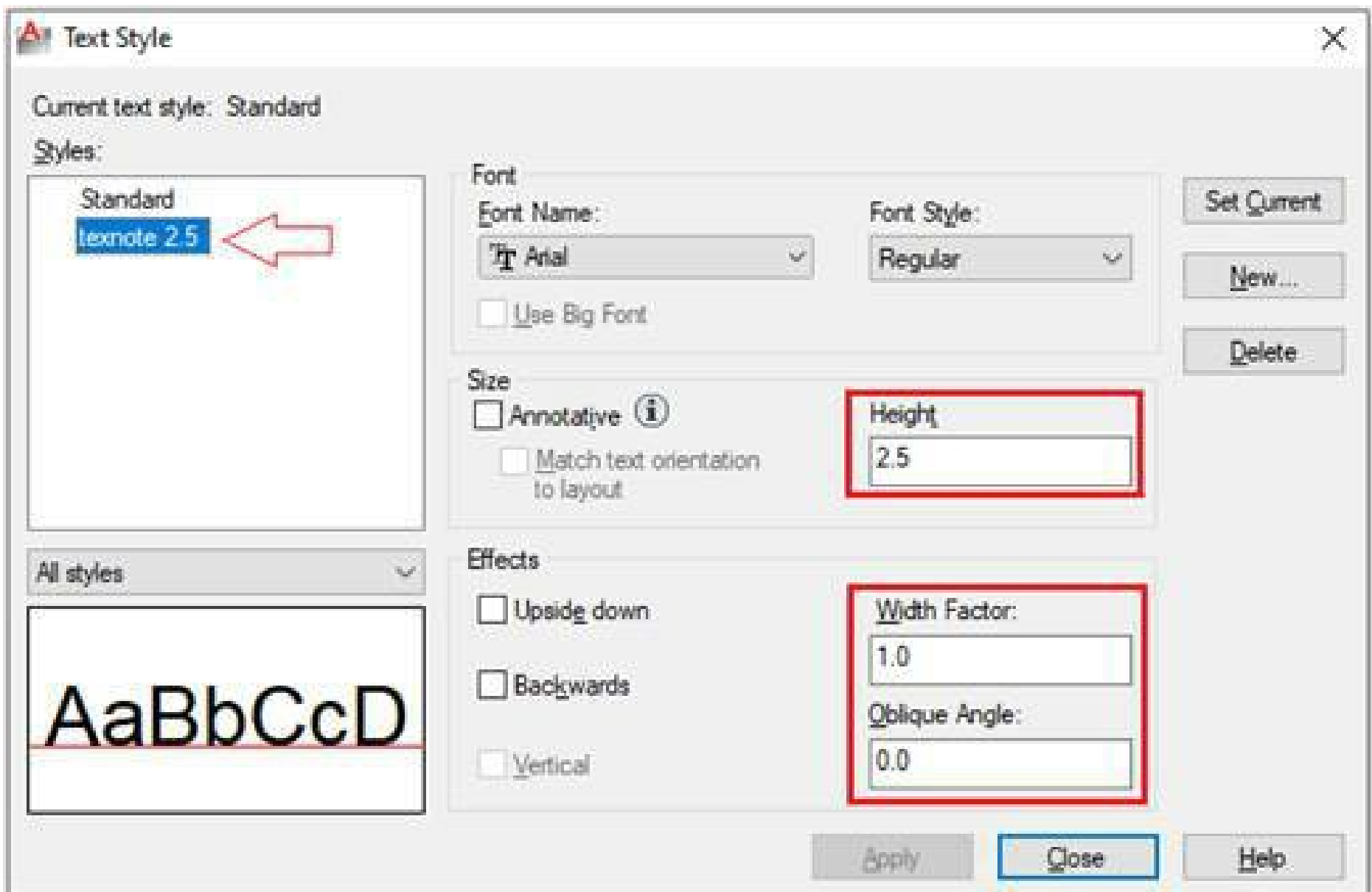
To create a new Text style, type the command: ST✓

You can choose *New* to create a new Text style or right-click an existing Text style and select **Rename** (don't choose the **Standard** style because it cannot be deleted and cannot be renamed).

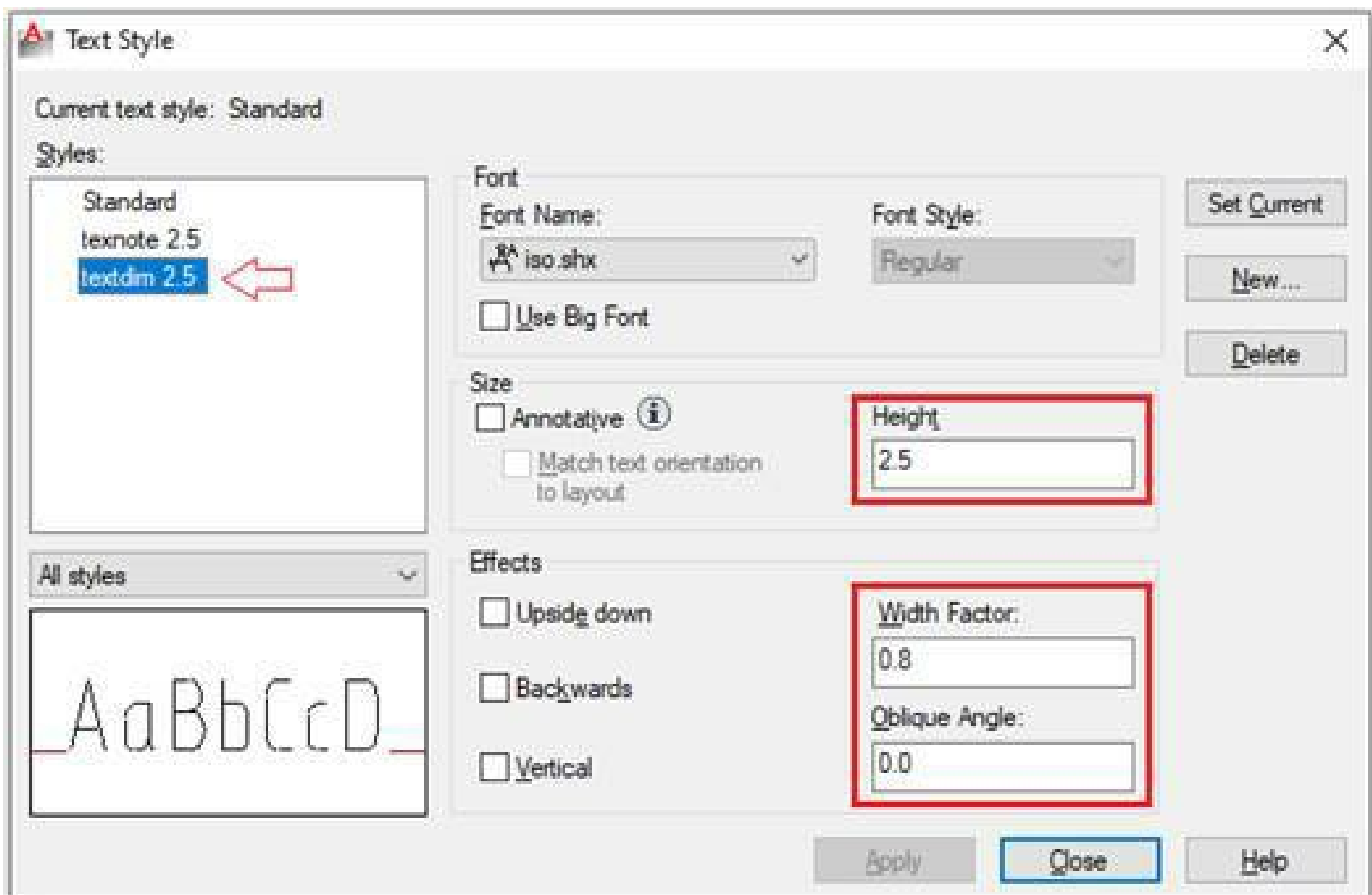


Name the new text style as “**textnote 2.5**”

Setting for “**textnote 2.5**”



Similarly, set another Text style is “**textdim 2.5**”



If it is A3 size: you can do the same with 2 styles: "textnote 3.5" and "textdim 3.5".

2.2. Should you use Annotative for Text style?

In Text style, there is an Annotative parameter - its use is to automatically zoom the text height according to the

drawing scale. It's not difficult to use, but I find it takes more time to edit than manually calculating the text height according to each scale.

Therefore, I do not use Annotative for Text style. We adhere to the principle from the beginning that, I do not impose on you, and you do not impose on me. I am excited to share my experiences.

I am not using Annotative for Text style but we will use Annotative for Dimstyle. Certainly so!

3. Write notes for drawings

Always keep in mind the following rule about the height of the note based on the drawing scale:

Text height on drawing in Model space = Basic height of text in paper size x (Drawing scale)⁻¹

Chapter 10: Is There A Way To Select Objects Faster?

2 Object Filter Tools

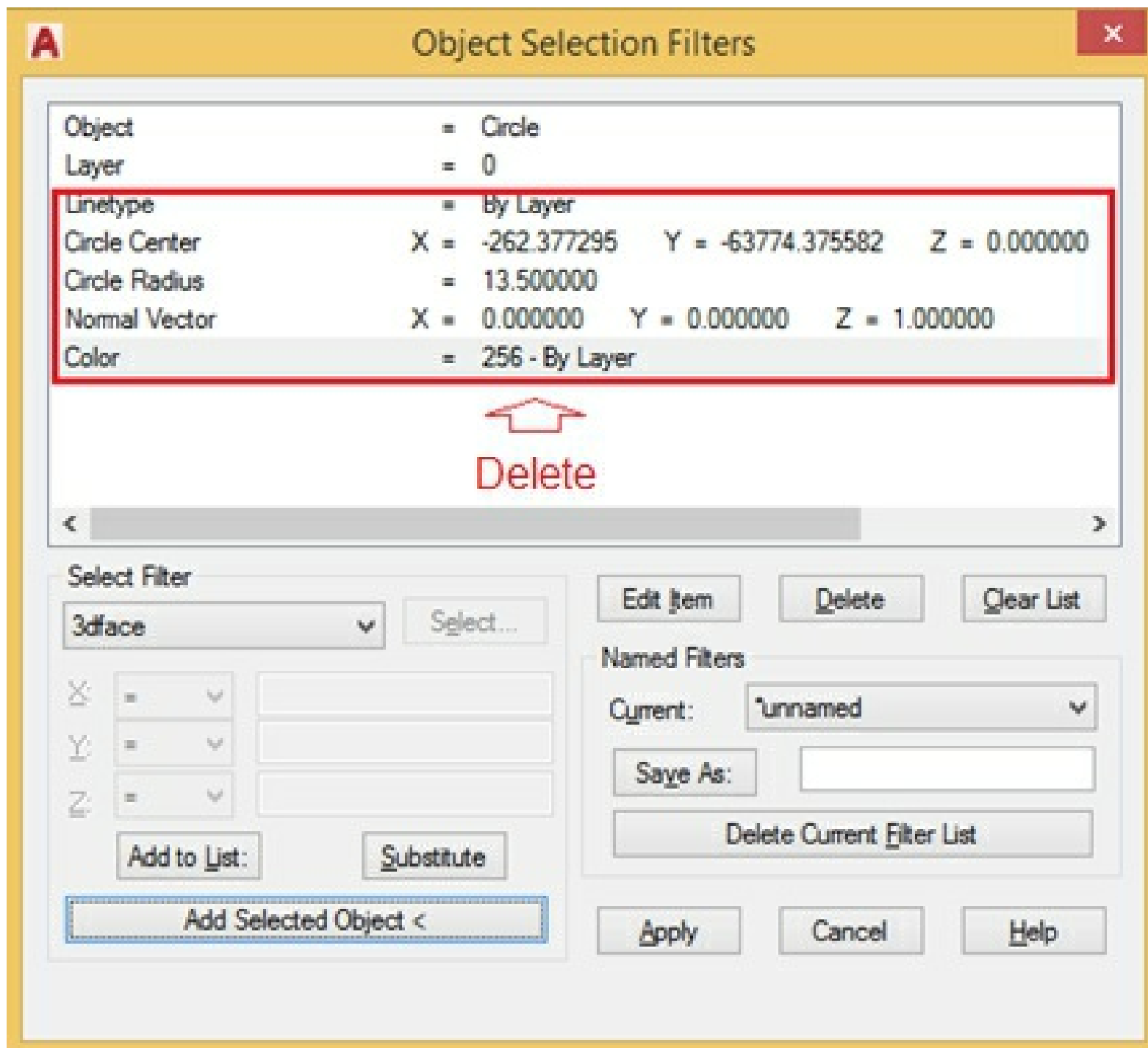
1. First filter tool: FILTER – Filter objects in the selected area

There are 2 ways to filter objects using Filter

1.1. Filter by sample object

The basic process of this method is:

- Type command: **FI**✓
- Click on Clear List to clear the history of the previous sample object
- Select Add select object
- Choose 1 object as a sample
- Delete all the private properties of the template object, keep only the common properties to filter by that common property.



- Apply

- Scan the drawing area we want to find the objects in it and press Enter!

--> All objects that share properties with the sample object are filtered out and selected. At this point, we use the next commands (Erase, Move, Copy, ...) to process the object according to our purposes.

1.2. Filter objects based on selected attributes in the first place

Instead of taking a sample object and then removing redundant properties, you can select those properties first and filter.

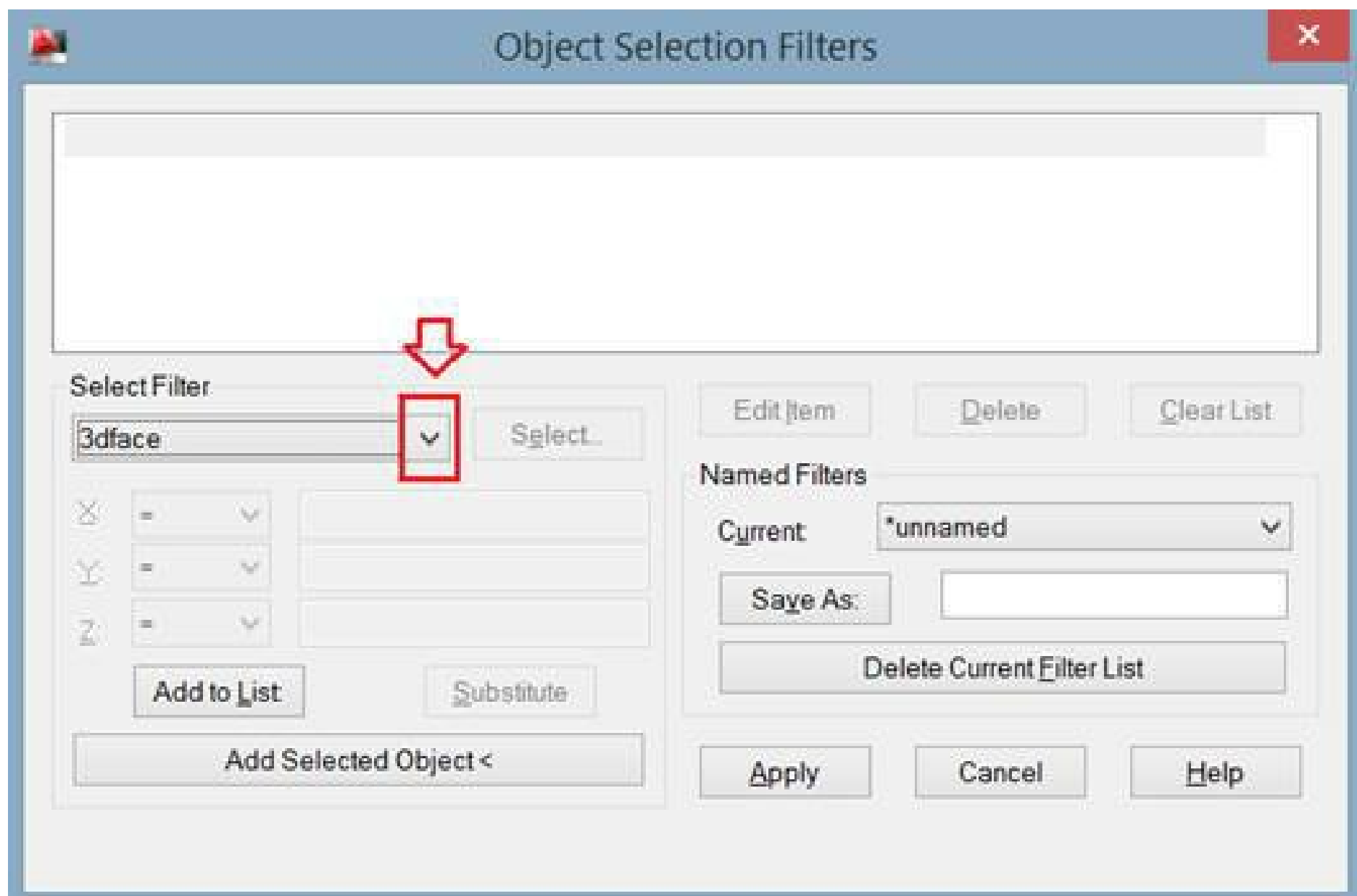
The basic process of this method is:

Select the properties to filter --> Add to List --> Apply --> Scan filter area --> Get filter results

- Type command: FI✓

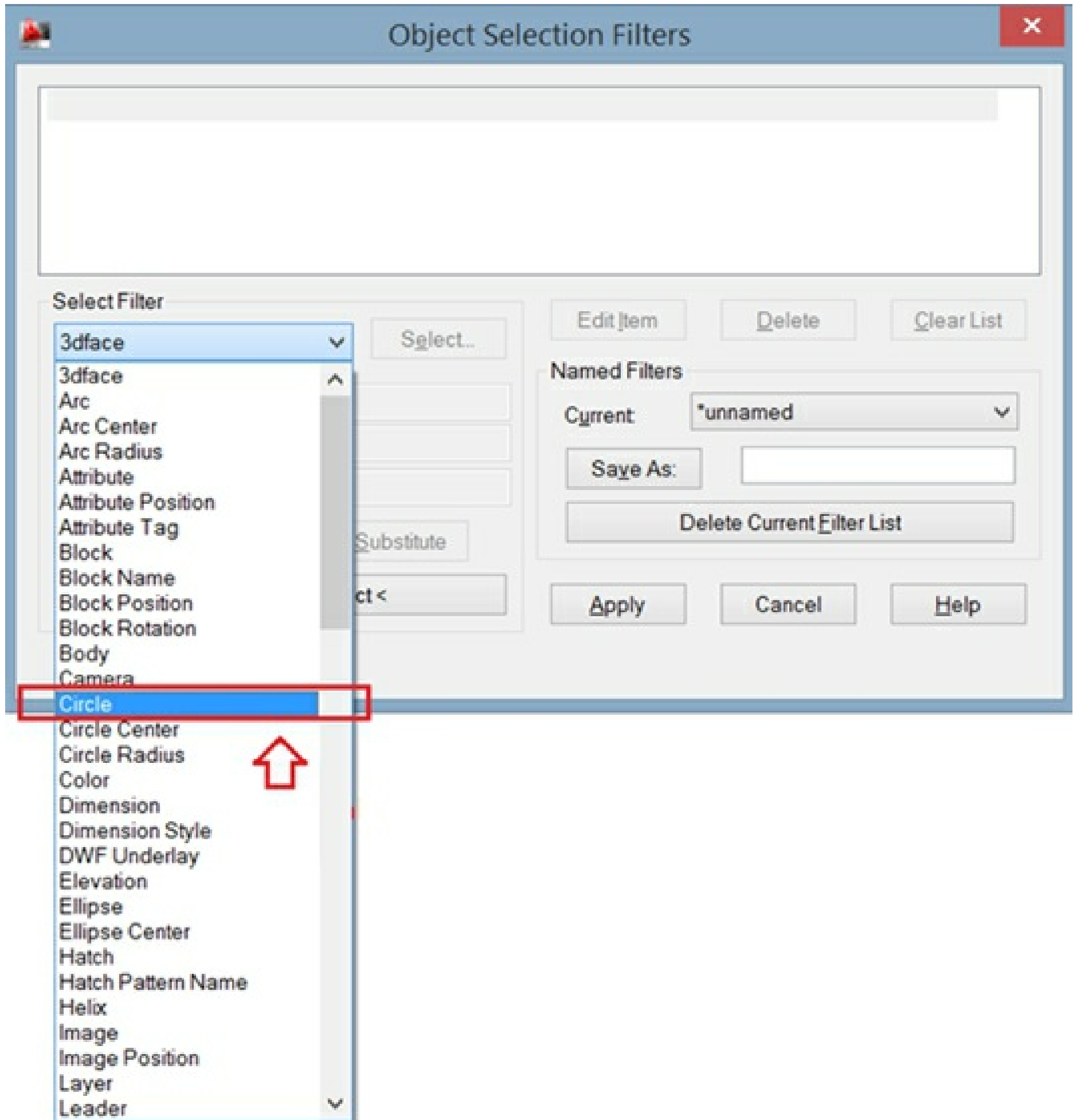
- Click on Clear List to clear the history of the previous sample object

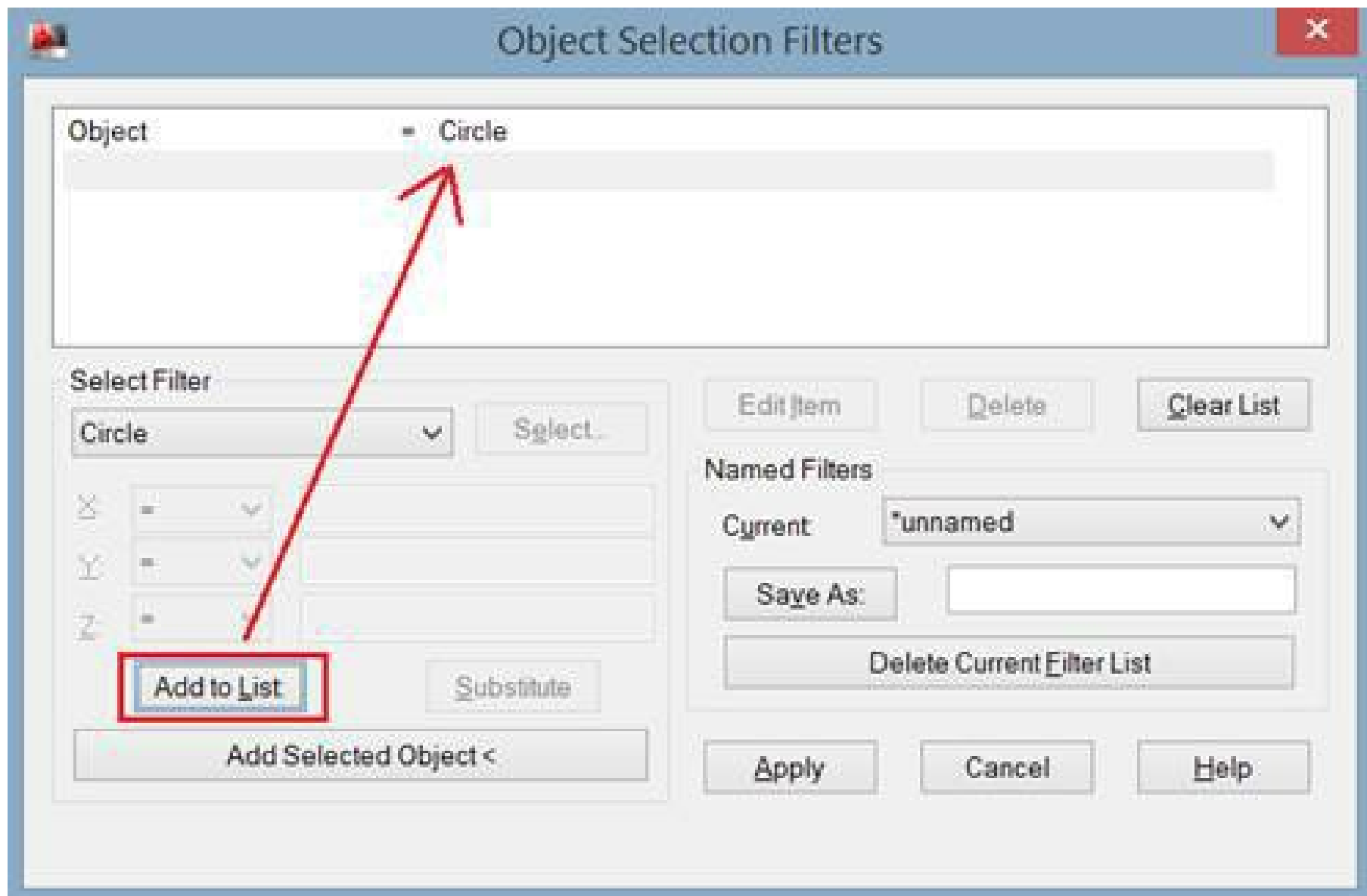
- Click the arrow in the Select Filter box and the Select button to select the attributes



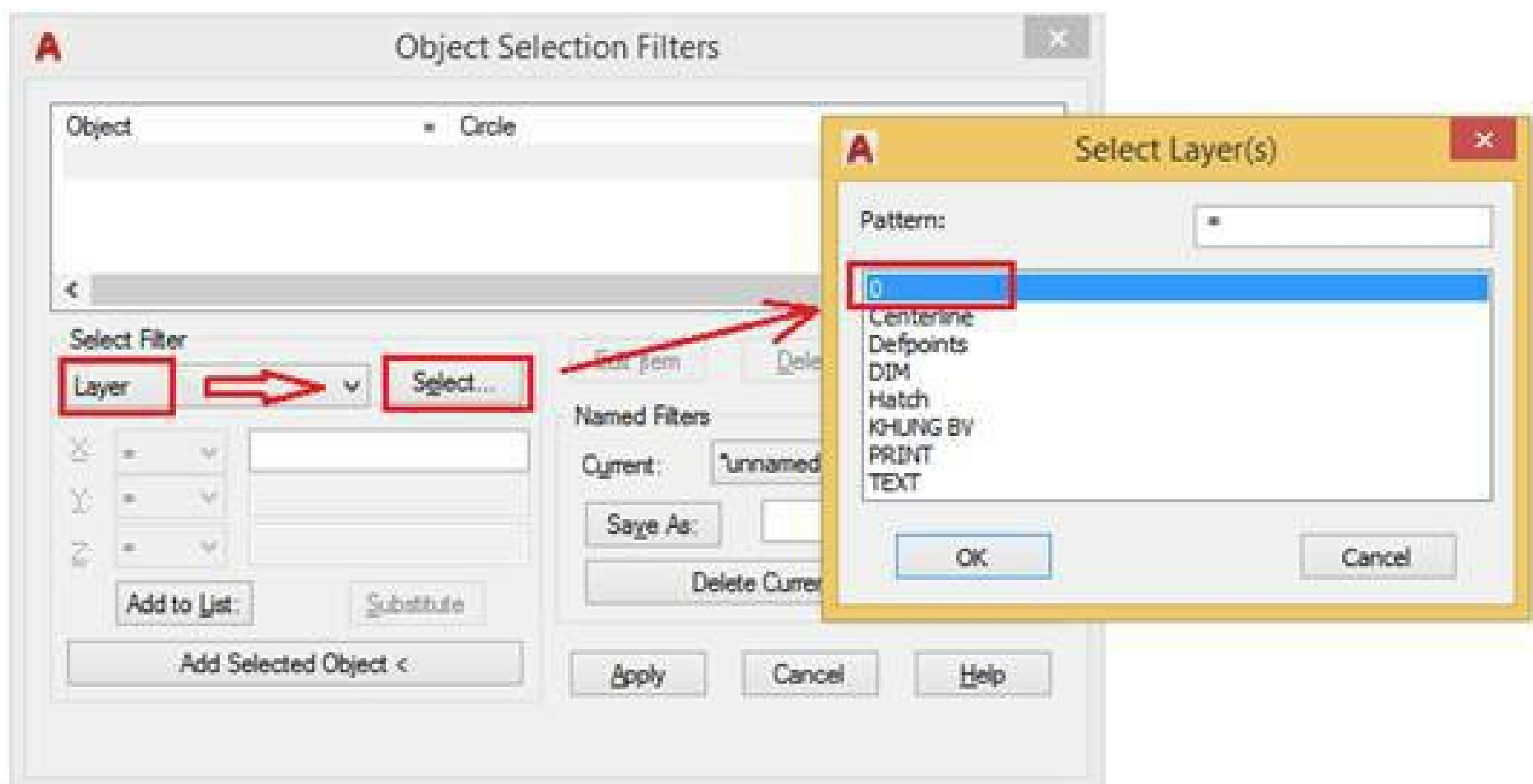
- Select 1 or more attributes you want to filter at the same time. Each time an attribute is selected, click **Add to List** to add it to the list of properties to filter.

Example: I want to filter out objects that are circles and belong to layer 0.

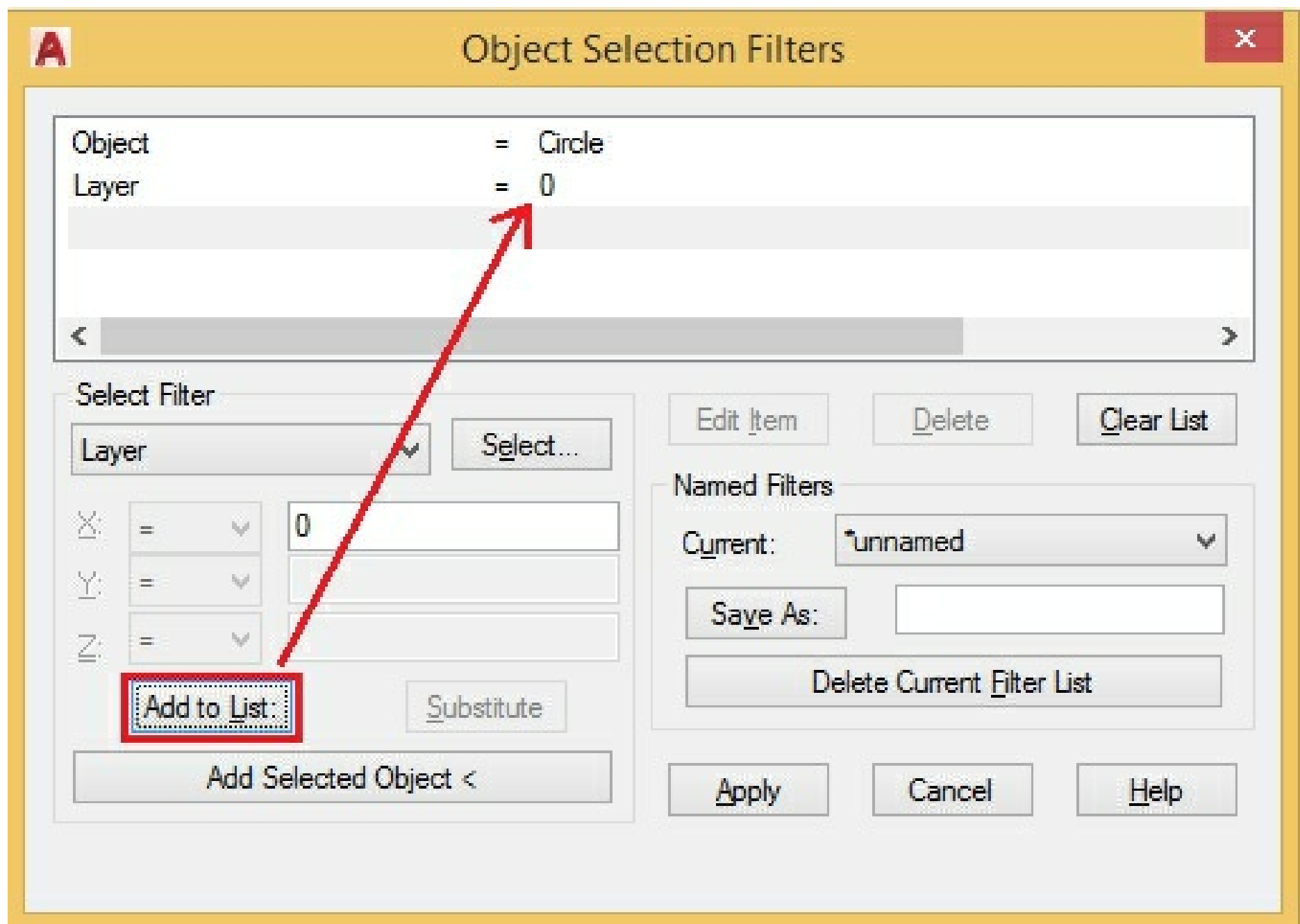




- The Circle property is included in the filter criteria.
- Select one more filter attribute as Layer / select Select / Select the name of the layer to filter is layer 0/ OK



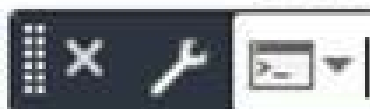
- Click **Add to List** to add to the list of filter properties



- Click **Apply** / Scan to select the drawing area you want to filter objects

Here I get a filter result of 1641 objects in the selected area satisfying "they are circles and belong to layer 0".

```
Command: FF FILTER Applying filter to selection.
Select objects: Specify opposite corner: 1641 found
Select objects: Exiting filtered selection.
```



Or you can see here:

Circle (1641)		CLI
Color	*VARIES*	
Layer	0	
Linetype	*VARIES*	
Center X	*VARIES*	
Center Y	*VARIES*	
Radius	*VARIES*	
Diameter	*VARIES*	
Circumference	*VARIES*	
Area	*VARIES*	

After you choose, what can you do? you can Move, Copy, Earse them.

With the Filter command, the way I usually use is way #1 – Filter by sample object.

How about you? Let me know your options.

Please write it down here:

.....

.....

2. Second filter tool: SELECTSIMILAR – Filter objects in the entire drawing space

Along with FILTER, SELECTSIMILAR is my favorite when it comes to filtering. I even use SELECTSIMILAR more than I use FILTER.

SELECTSIMILAR is a command that searches for objects in the entire drawing space, based on predefined attributes.

- Type command: **SELECTSIMILAR**✓
- At the prompt *Select objects*: you type SE✓ to go to the Settings to select filter properties



Table of filter properties

If you want to search for objects by any common attribute, check the box and then click OK.

- Select 1 object as a sample object to search
- > Objects that share the same properties (according to the properties to be searched) are selected.

***Note:**

- We don't always have to go to the Setting dialog, we only go here when we need to change the search property, it will automatically save for the next command execution.
- SELECTSIMILAR clearly distinguishes Mtext and Text
- SELECTSIMILAR searches for Color and Linetype properties and distinguishes them according to their features: By Layer, By Block

Chapter 11: I Want Even Faster!

Using the filter tools, I easily selected thousands of objects in less than 10 seconds. I use the Move command to move them some distance. However, right after that, I found it unreasonable. I need to move them one more paragraph to be exact.

What should I do, if I don't want to waste my time filtering the object again like it was at the beginning?

Method 1: Perform the re-selection using 1 of 2 filter tools

Method 2: Another secret for you. A trick but very useful. It is a knife in the sleeve.

PREVIOUS_ Re-select the object that was edited just before.

If you have just edited (Move, Copy, Scale, Earse, Trim, Extend, ...) one or more objects (even if it's a mess). Right after that, you want to make further corrections to them.

- Redo the command you want to edit (Move, Copy, Scale, Earse, Trim, Extend, ...)
- At the prompt: Select Objects prompt, type command: **P** ✓
- > the whole object you made earlier is called again.
- Press Enter to finish selecting them again.

Too fast, too dangerous, isn't it!

Try it!

Chapter 12: Dimstyle Management - A Method Of Noting Dimensions On Technical Drawings

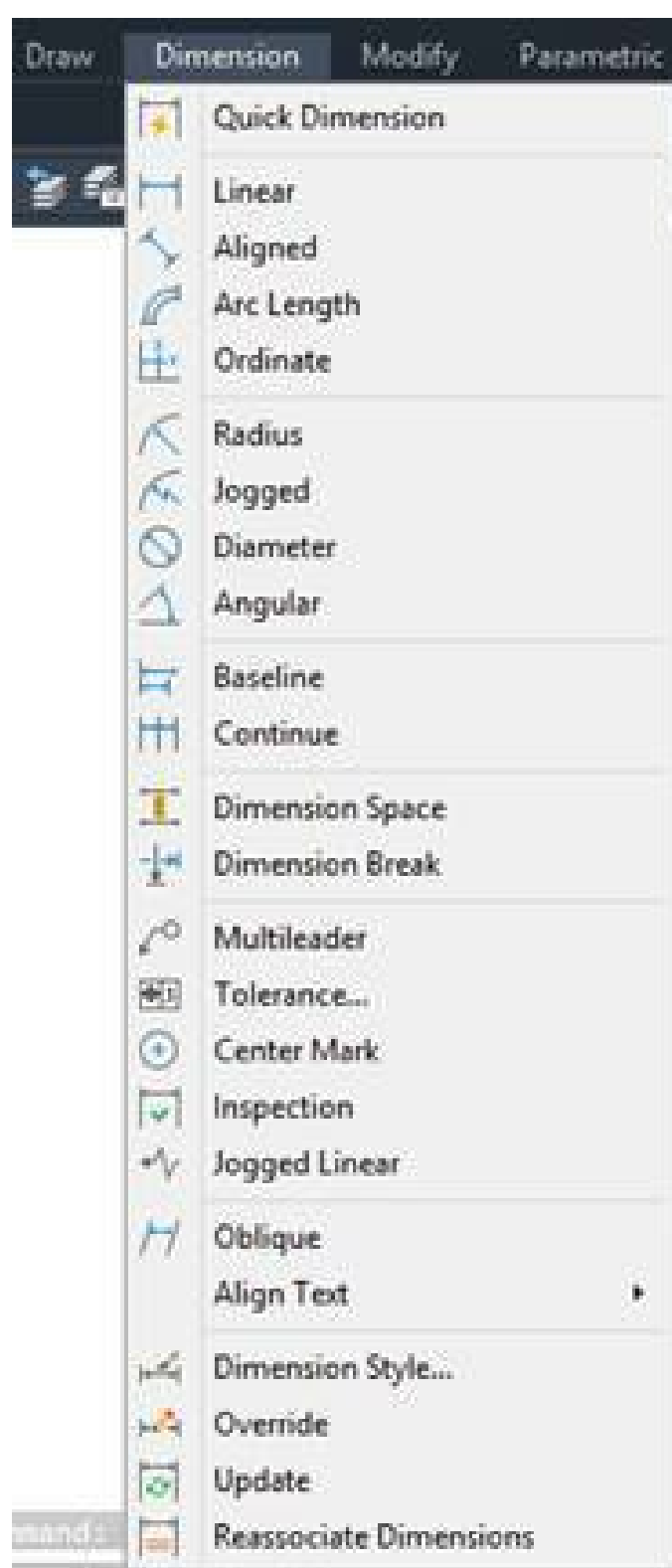
This book will only be for those who build 3D models on certain software, then export the projection data to AutoCAD and present and print 2D drawings on AutoCAD software. The object of the application is very clear! If you are not this person or find it unnecessary then I suggest that you close this book now, because it is not suitable for you. And if you're still interested, I'm always excited to share everything I've experienced.

If you only use CAD to note the dimensions, it sounds dry right?

But don't think so. There is a whole art to bring complicated, dry things into simple, subtle things waiting for you ahead.

1. Dimension commands

There is a series of dimensioning commands in AutoCAD. It's on the Dimensions tab.



I have only listed some of the most commonly used commands, but I will not describe how to use them, because they are familiar to you. This is not a book for beginners.

- DLI - Command to write dimensions in the perpendicular direction
- DRA - Command to write the size of the radius
- DDI - Command to write dimensions of diameter

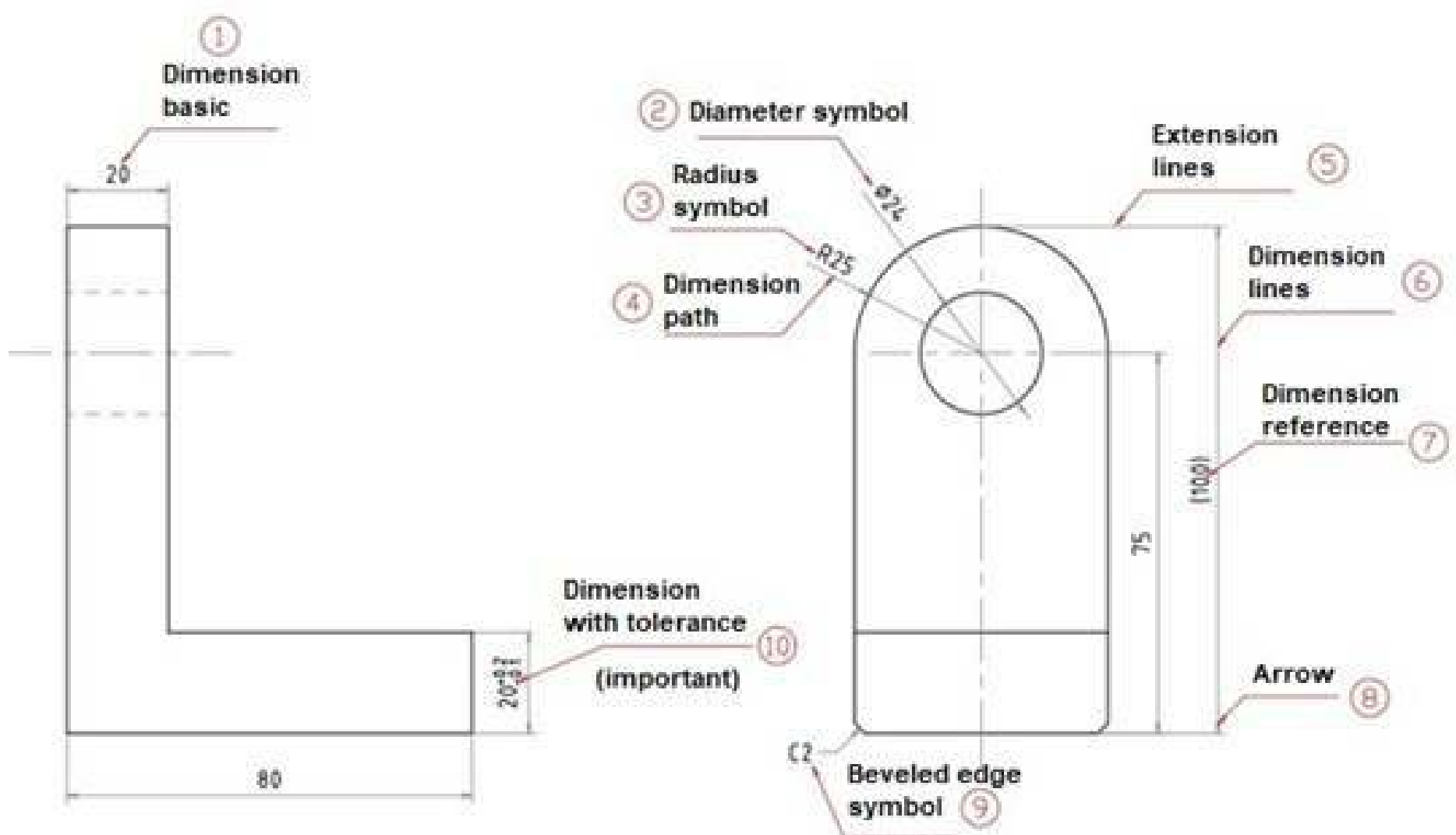
- DCO - Command to note dimensions continuously

For those of you who do mold design, also use the DIMORDINATE command.

2. Regulations on how to note dimension

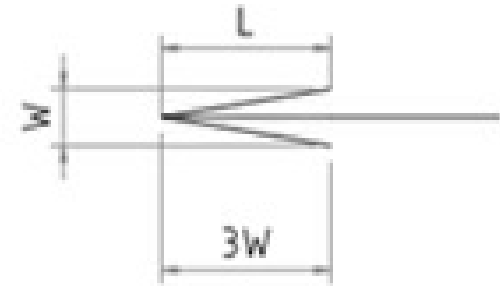
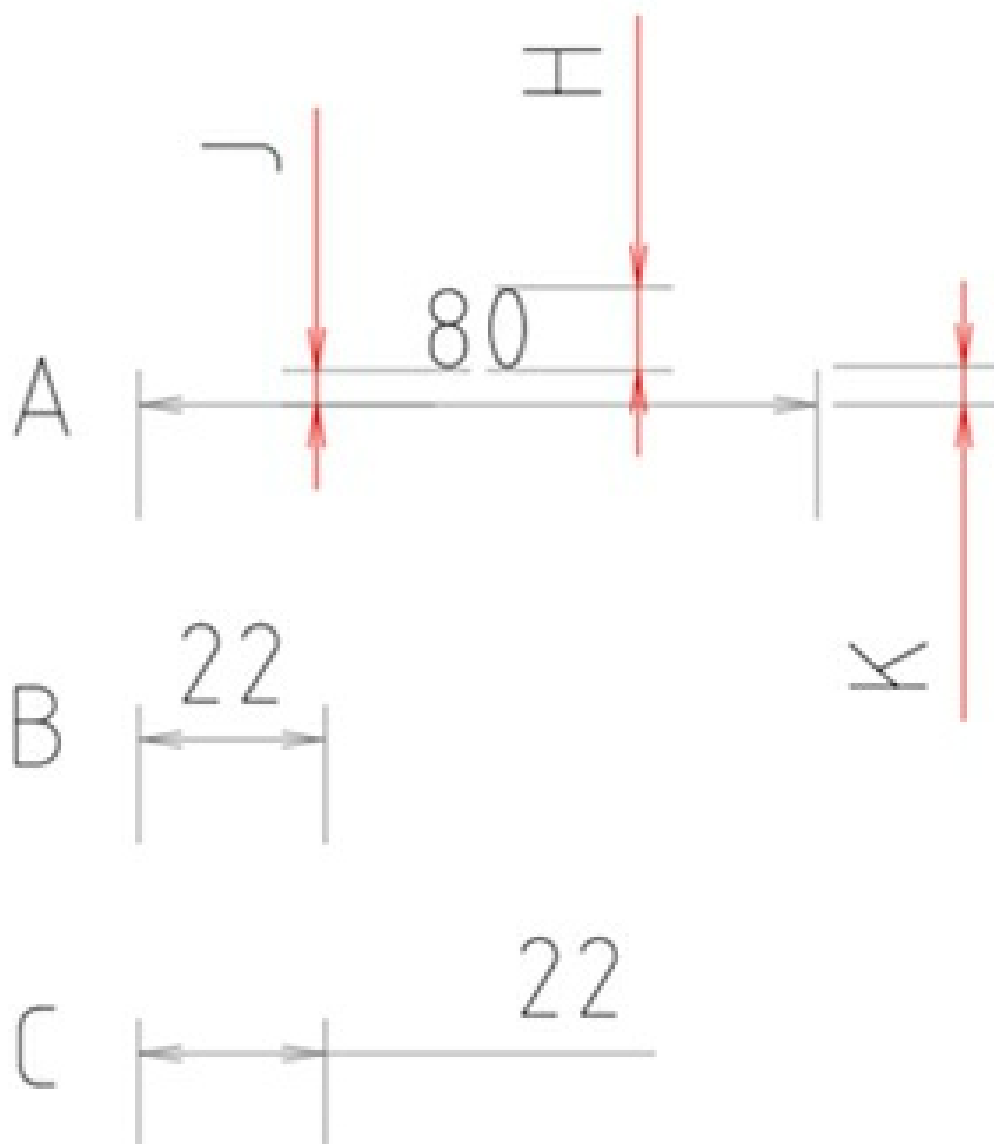
2.1. General rules when noting dimension

- The font used to record the size is Font: iso.shx or isocp.shx. The height of the dim is equal to the base height of the text
- Each part dimension is recorded only once on the drawing (including all views)
- Dimensions should be indicated on the view that depicts the most detailed shape
- On drawings, the convention is to only record dimensions for angles other than 90 degrees, 90 degrees do not need to be recorded
- Dimensions should be written outside the contour of the part
- Dimension lines should be aligned and should be in groups
- Crossing dimension lines should be avoided. In the case of mandatory intersection, there is no need to interrupt.
- The minimum distance between the first dimension line and the part is 10mm
- The distance between adjacent parallel dimension lines is at least 6mm
- Distance between dimension line and part is 0
- The dimension line must extend past the dimension line by a minimum distance of 1.5mm.
- The path used to dimension arcs and circles must be radial
- When measuring a circle or arc, its center must be specified by a "+" centerline.



2.2. Rules for arrows and numbers

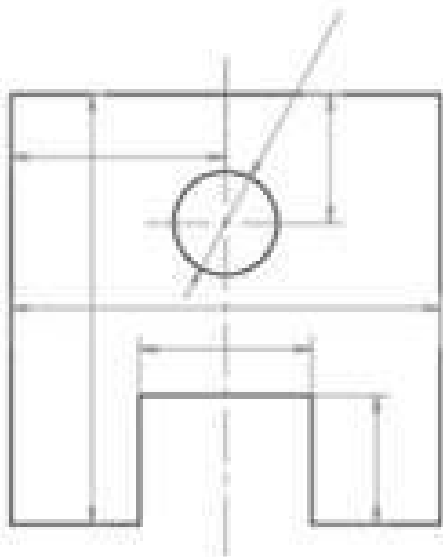
- Arrow style: Open style
- Arrow parameters are specified to apply to each paper size



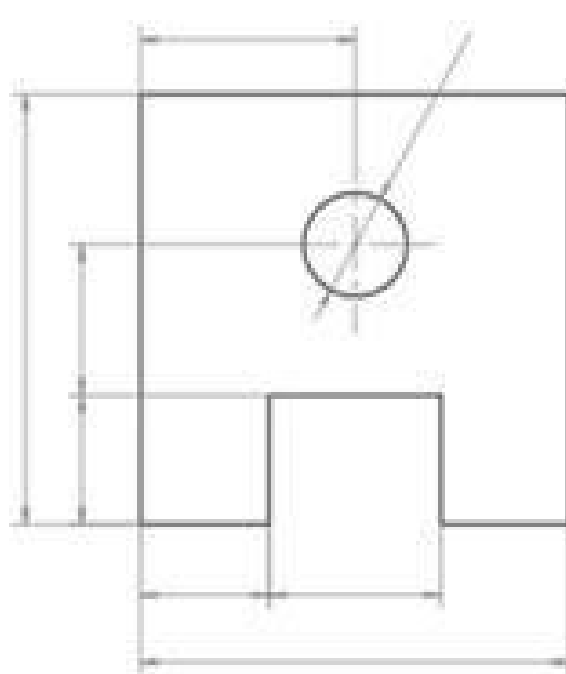
Size	W	H	J	K	L
A4	0.5	2.5	1	1	1.5
A3	0.7	3.5	1.5	1.5	2
A2	0.8	5	2	2	2.5
A1	1.2	7	3	3	3.5
A0	1.6	10	4	4	5

2.3. Specify the layout and presentation of dimension lines

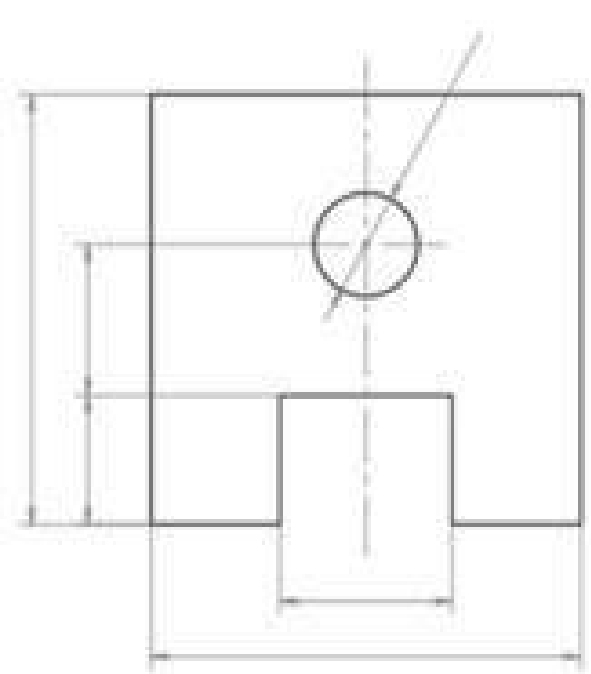
- Dimension lines are drawn outside the contour of the object (B)
- Do not use more than 3 dimension lines on one side of the object (B)
- For symmetric objects, the centerline will pass through the midpoint of the dimension line (C).
- When it is mandatory to use many parallel dimension lines in the drawing, then:
 - ü Arrange the size lines in ascending order from inside to outside
 - ü The size numbers should be staggered to ensure the distance between the size lines



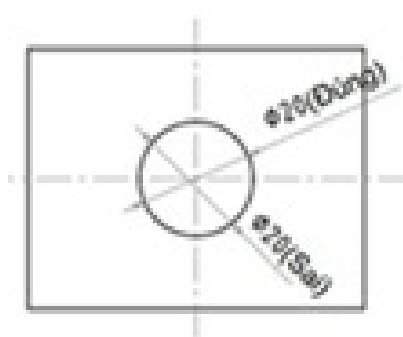
A (FAIL)



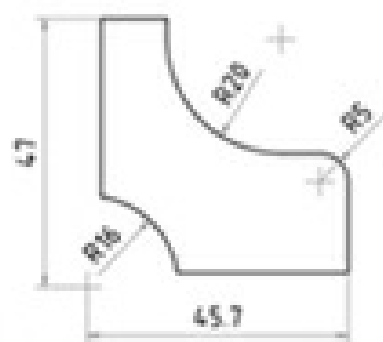
B (TRUE)



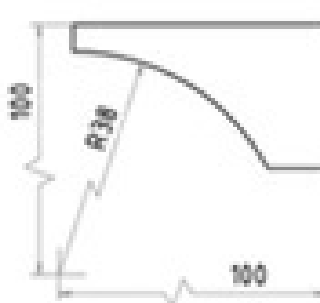
C (TRUE)



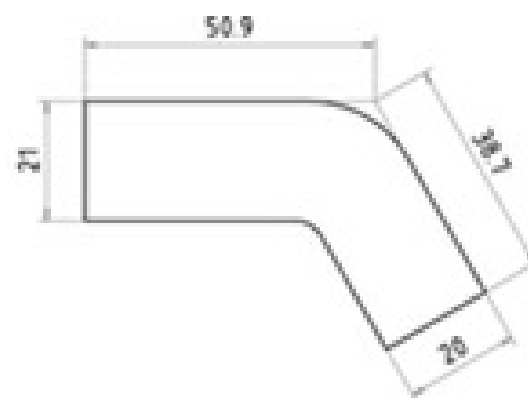
The guide used to note the dimensions for the hole diameter must be radial



If the arc is less than 180 degrees, note the size of the radius - the R symbol is placed before the numeric value



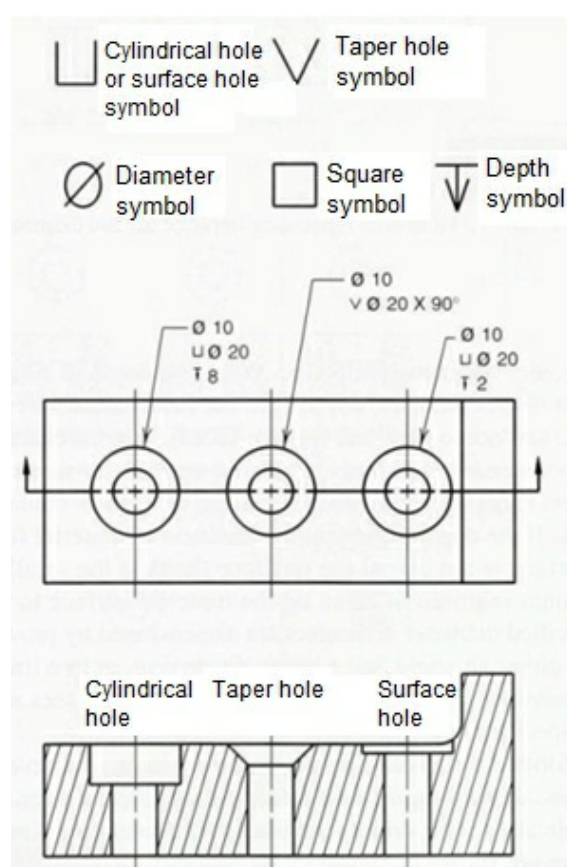
Use a shortened line for arcs whose centers are outside the drawing scope



When positioning a point by intersecting guides, the dimension line must also pass through that point

2.4. Regulations on hole dimension notes

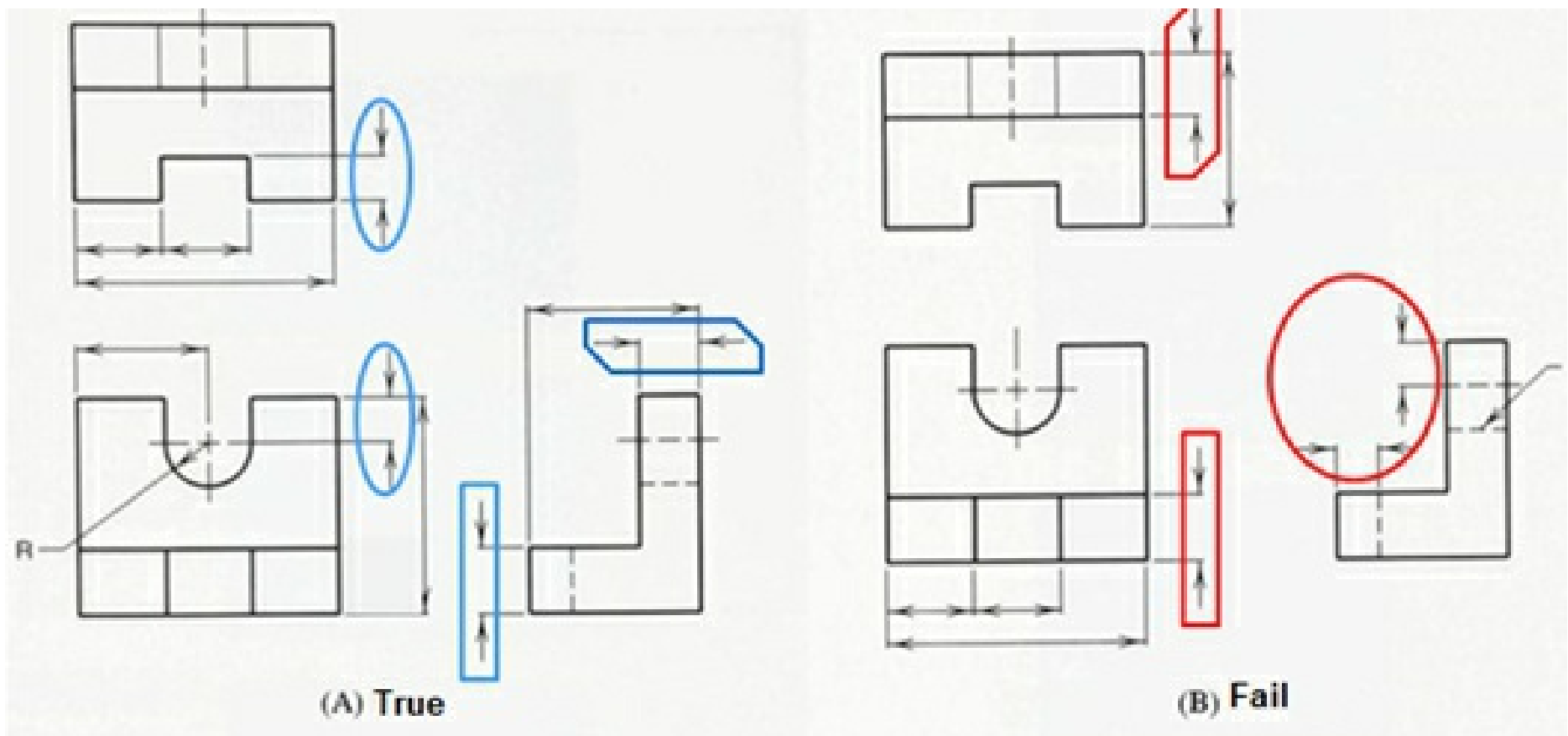
- Dimensions should be indicated on the projection that best describes the shape of the hole.
- Dimensional path must be radial
- The diameter symbol must precede the numeric value.
- Cylindrical, taper, and facet bore all have their designation, which shall precede the diameter designation.
- The depth symbol precedes the size digit indicating the depth of the hole.
- When the hole is through, there is no need for a depth symbol.



3. Methods of dimension notes

3.1. Method for noting dimensions by contour

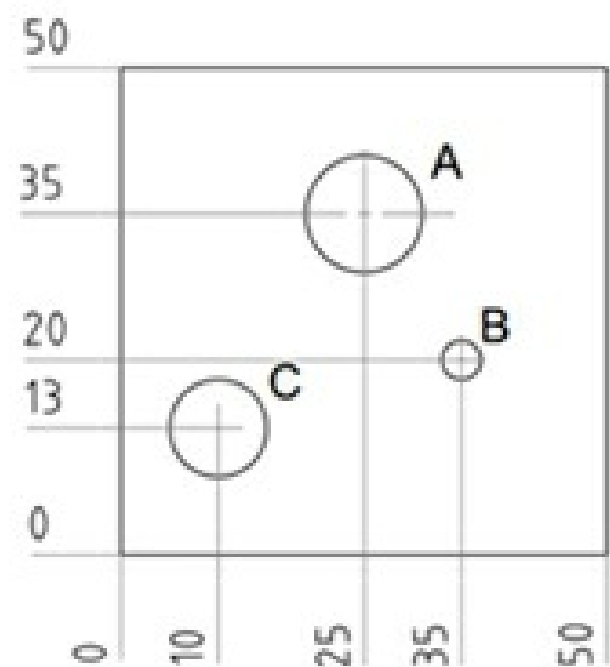
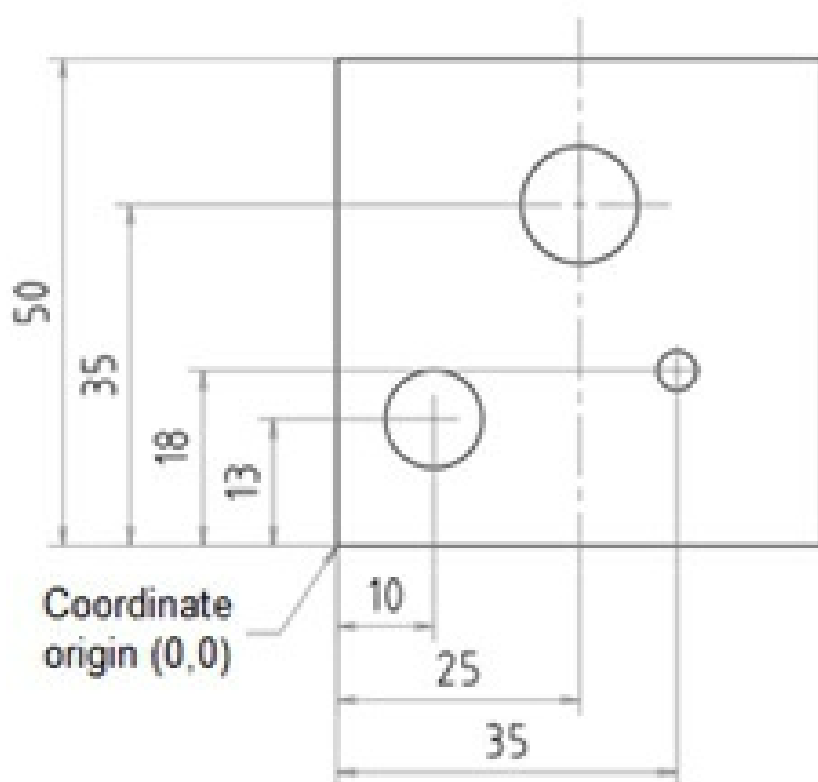
- When noting the dimension, it is advisable to choose the position of the dimension note that best shows the geometrical characteristics of the object.
- When the design part is machined only once, there is an assembly standard in the center of the part, of the part is locked.



3.2. Method of noting dimensions to the origin (Absolute)

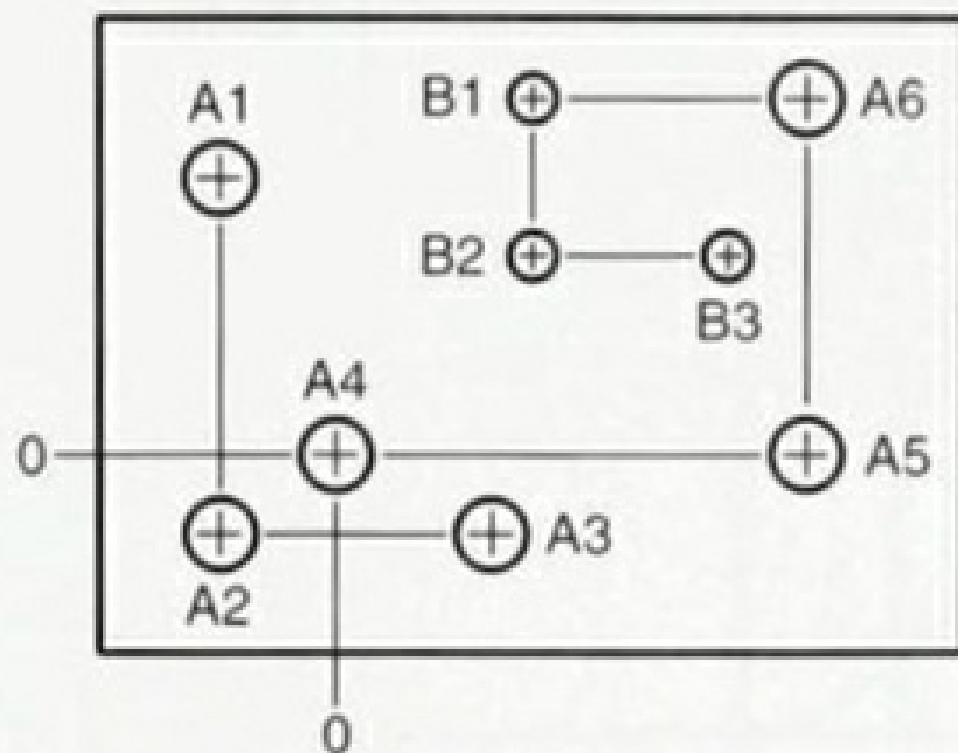
- This method has the effect of avoiding errors due to dimension accumulation.
- When the design details use processing technology on machine tools: milling, grinding, turning ... or standard assembly at corners and edges.

Symbol	A	B	C
Diameter hole	12	10	4



3.3. The method of noting dimensions according to the coordinate table

- This method reduces the number of dimension lines
- Use when the drawing has many points that need to be located



Hole	X	Y	Symbol
A1	-1.00	2.00	Ø.50
A2	-1.00	-.50	Ø.50
A3	1.10	-.50	Ø.50
A4	0	0	Ø.50
A5	3.38	0	Ø.50
A6	3.38	2.62	Ø.50
B1	1.50	2.62	Ø.25
B2	1.50	1.50	Ø.25
B3	2.88	1.50	Ø.25

Chapter 13: The Art Of Noting The Dimension On The Drawing

This will also be one of the most important contents of the book. This is the part I want you to apply immediately and always in your current work.

It remains a secret to most of the mechanical engineers out there, even to those with decades of experience working with AutoCAD. They don't know!

It is worth mentioning that this method is extremely simple.

But extremely useful!

Perhaps turning complex things into simplicity is an art.

Actually!

Let me ask you a question

Are you doing drawings with a lot of scales and have to create a lot of Dimstyles?

I know exactly what you're doing. For each drawing scale

You will create a Dimstyle, in which you will edit the corresponding Use overall scale of for each drawing scale.

Forget about that!

There's no rule limiting the number of Dimstyles per drawing, but the more Dimstyles, the heavier the drawing will be.

And if I help you just use a single Dimstyle for all types of drawing scales, what do you think?

Are you interested?

What is the secret here?

It's *ANNOTATIVE*!

- With Annotative, you only need to use a single Dimstyle for all types of drawing scales (as is, zoom in or out).
- However, it does not apply to drawing areas that use detail area: for each "real draw scale" you should create a separate Dimstyle for them.

You may be wondering:

If used a Dimstyle for all drawing scale types. So why can't we use that same Dimstyle to detail area?

Answer:

Okay. OK. I used to do the same thing a lot. But it won't be safe.

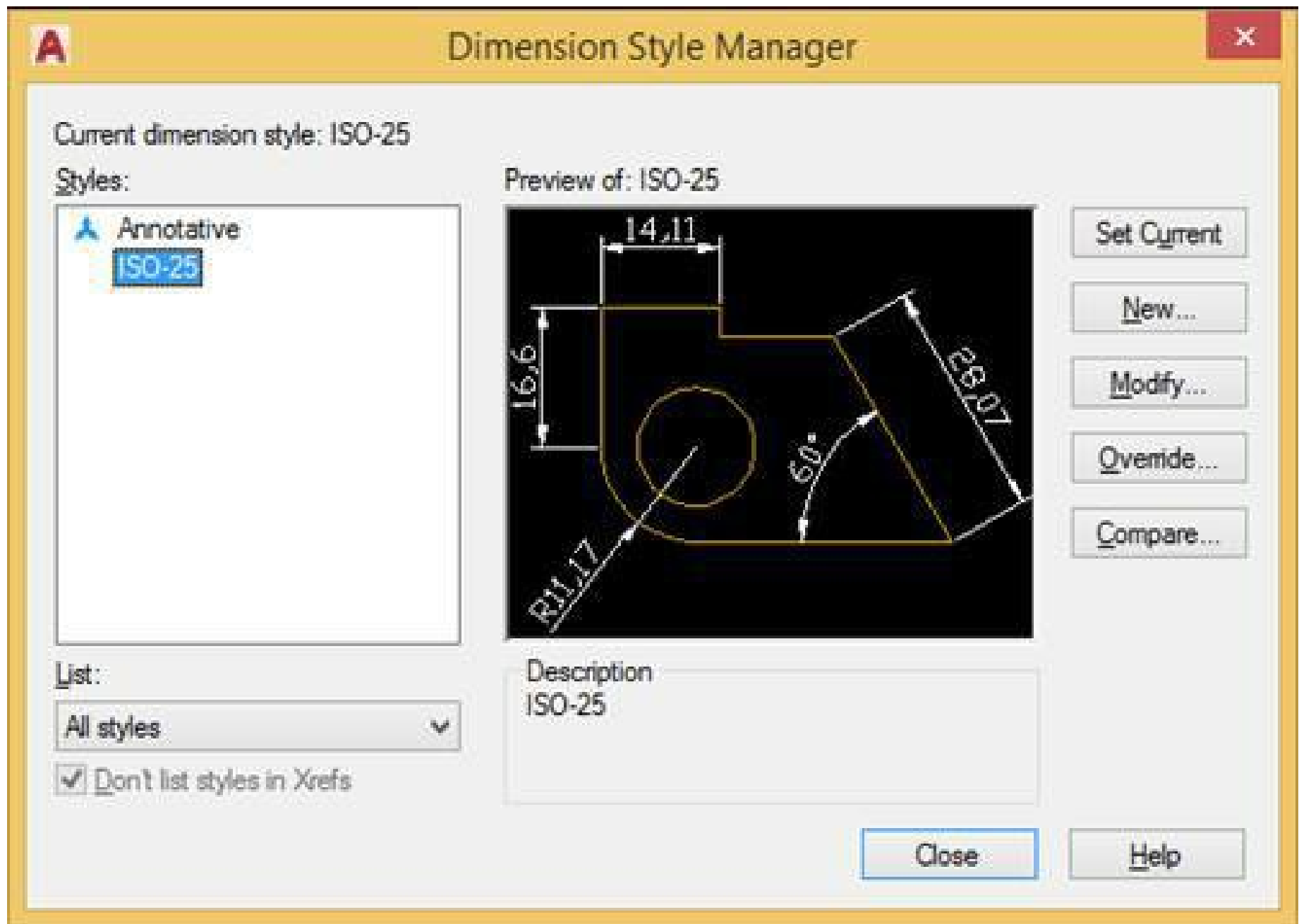
I will give you a complete and convincing answer in the next chapter.

Be assured. I just give you the best advice!

1. Create and manage DIMSTYLE

We will only need to create a single Dimstyle with a "real draw scale" of 1-1. And you have to assign an Annotative to this Dimstyle.

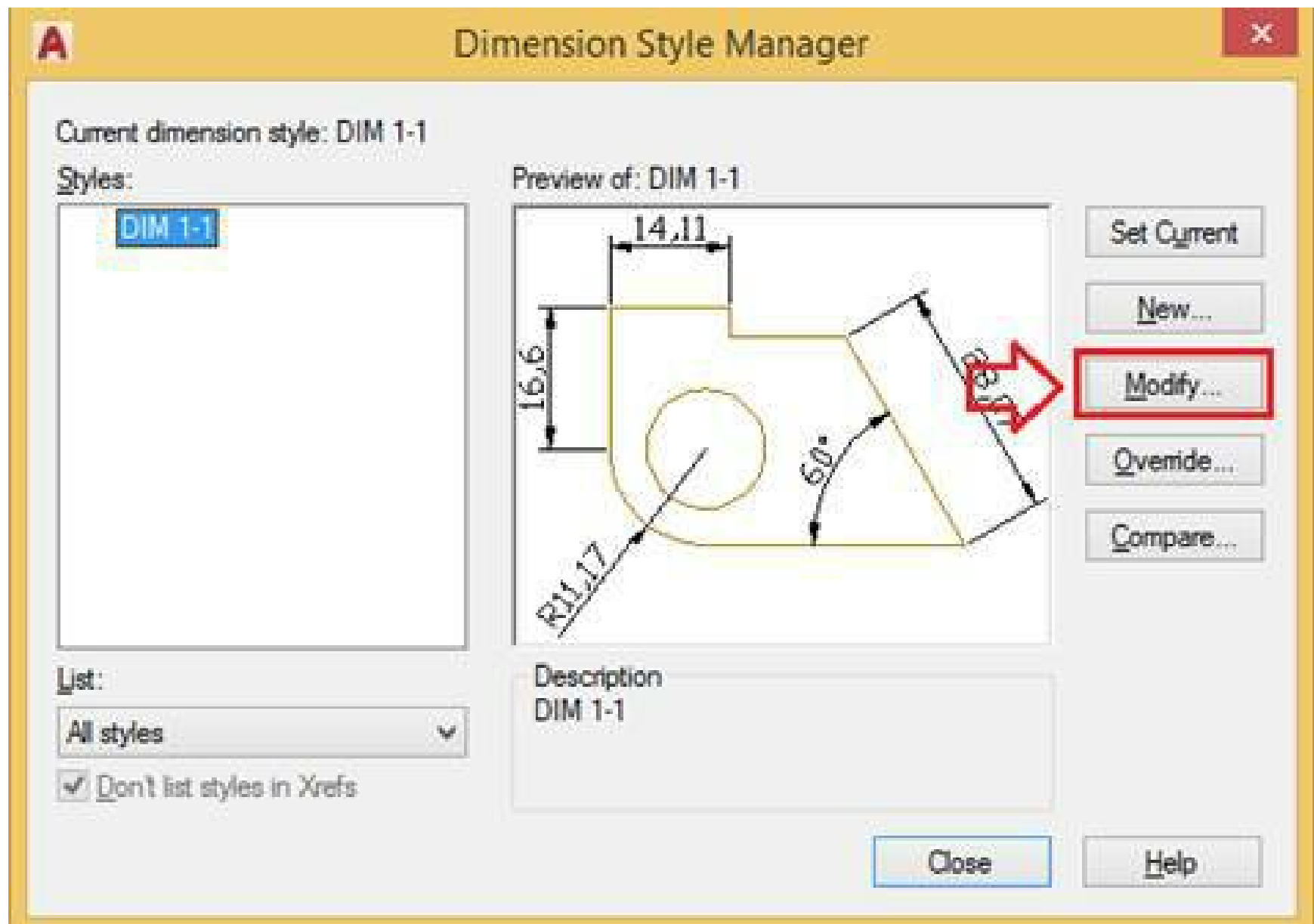
- Type the command: D✓



- Delete 1 existing Dimstyle of AutoCAD, keep only any 1 Dimstyle.

Let's say I keep ISO-25

- Rename this Dimstyle to the name you want, I usually use the Dimstyle name "**DIM 1-1**" (if there is more than 1 type of paper size, you can set details to distinguish by paper size, for example, DIM A4 1-1, DIM A3 1-1, ...)
- Click *Modify* to proceed with the installation



Install tabs inside:

A

Modify Dimension Style: DIM 1-1

X

Lines Symbols and Arrows Text Fit Primary Units Alternate Units Tolerances

Dimension lines

Color:

ByLayer

Linetype:

ByLayer

Lineweight:

ByLayer

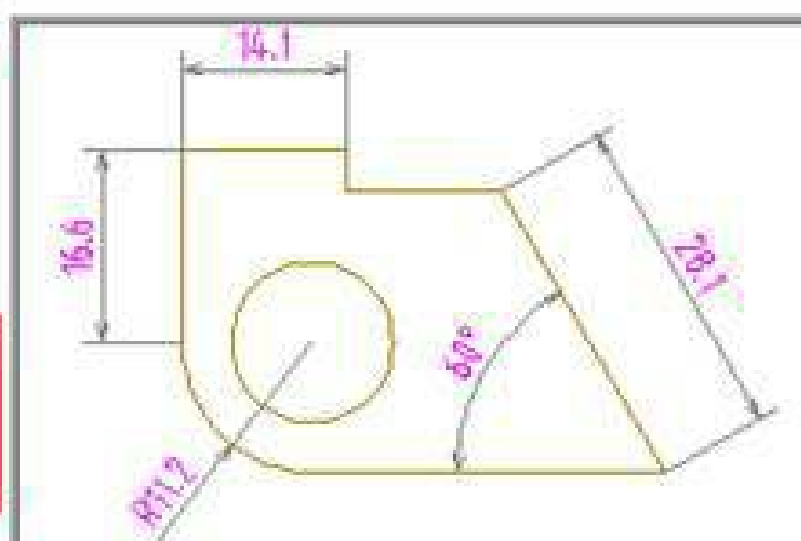
Extend beyond ticks:

0

Baseline spacing:

7

Suppress:

☐ Dim line 1☐ Dim line 2

Extension lines

Color:

ByLayer

Linetype ext line 1:

ByLayer

Linetype ext line 2:

ByLayer

Lineweight:

ByLayer

Suppress:

☐ Ext line 1☐ Ext line 2

Extend beyond dim lines:

1.5

Offset from origin:

0

☐ Fixed length extension lines

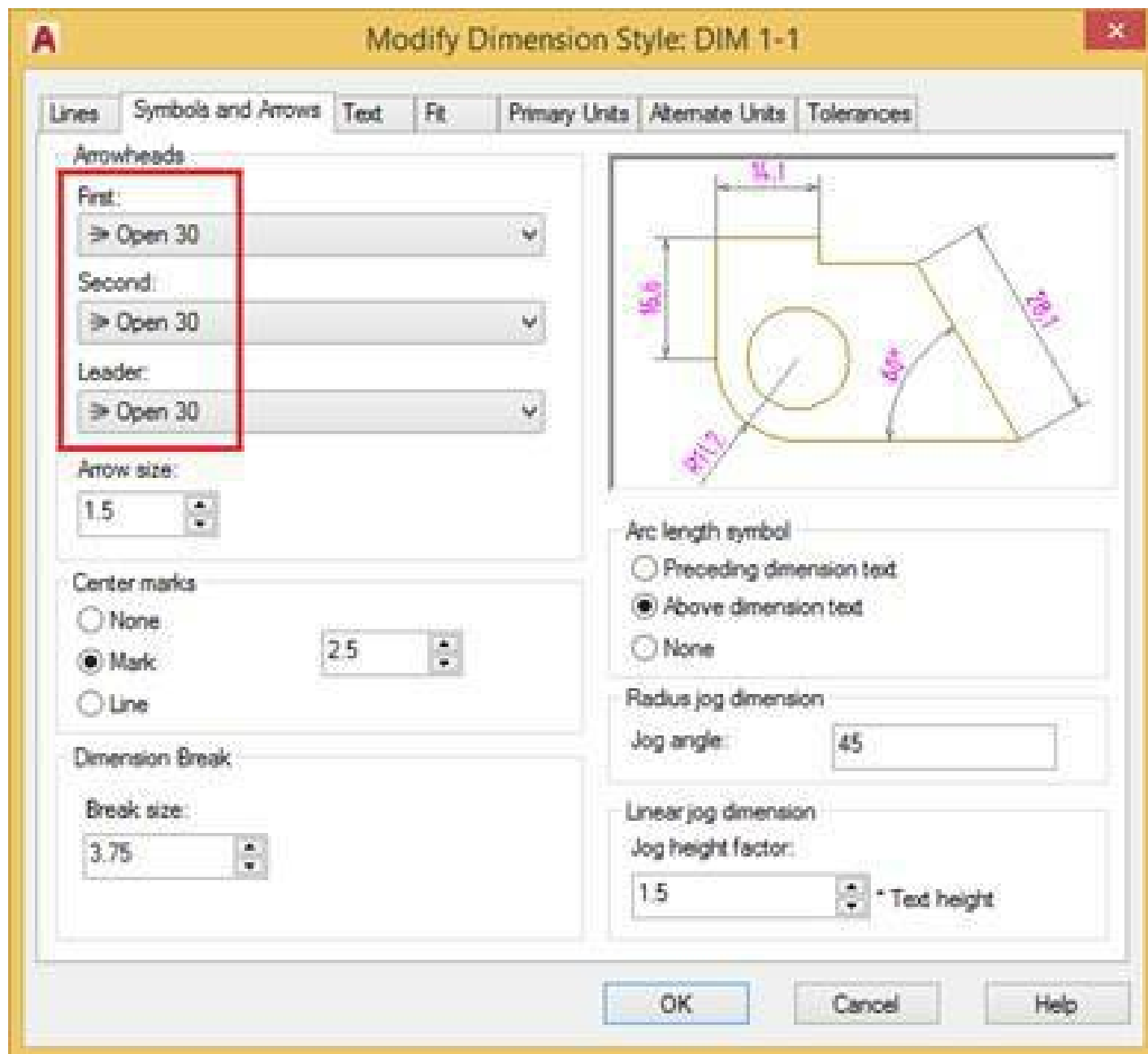
Length:

0

OK

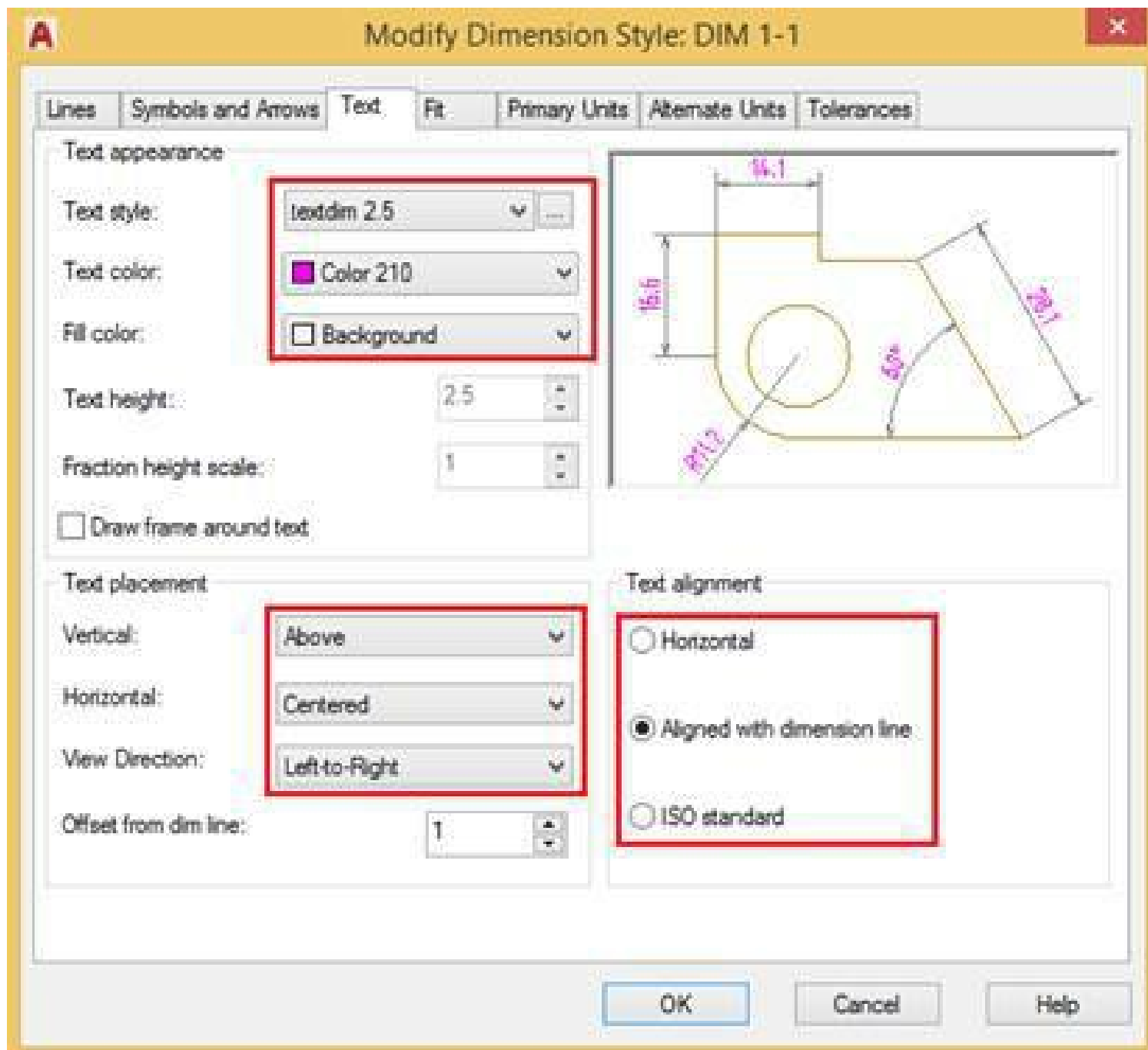
Cancel

Help



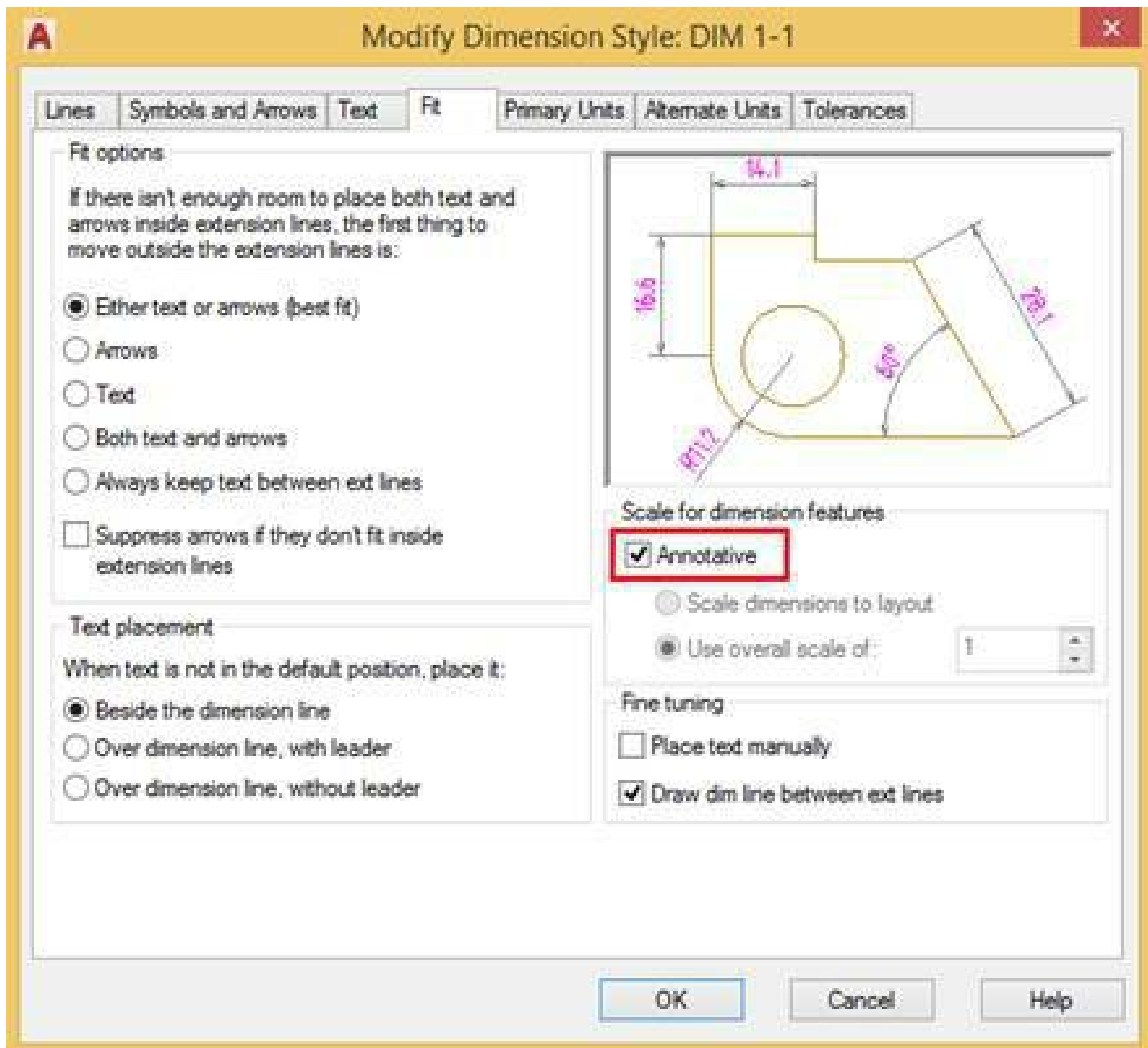
In the Text tab

- Select Textstyle as textdim 2.5
- Color: choose font color for size digits (I choose 210)
- Other contents: install as shown



But the biggest secret is in the **Fit** tab.

Please check the **Annotative** box in the **Fit** tab.



And always keep the default *Scale factor* = 1 in the **Primary Units** tab

A Modify Dimension Style: DIM 1-1

Lines Symbols and Arrows Text Fit Primary Units **Alternate Units** Tolerances

Linear dimensions

Unit format: Decimal

Precision: 0.0

Fraction format: Horizontal

Decimal separator: . (Period)

Round off: 0

Prefix:

Suffix:

Measurement scale

Scale factor: 1

☐ Apply to layout dimensions only

Zero suppression

☐ Leading ☒ Trailing

Sub-units factor: 100

Sub-unit suffix:

☐ 0 feet ☐ 0 inches

Angular dimensions

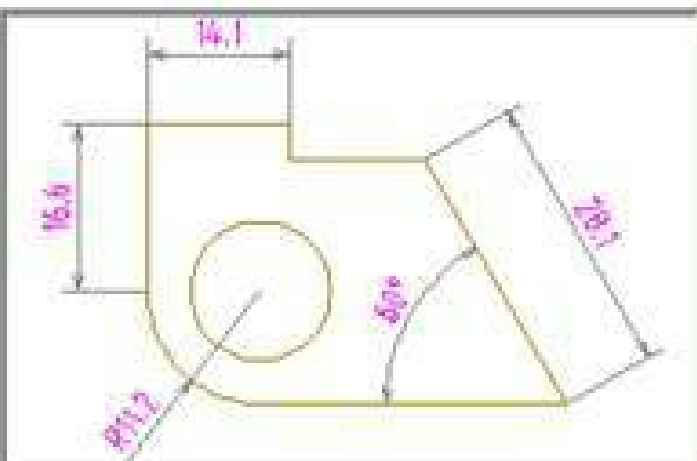
Units format: Decimal Degrees

Precision: 0.0

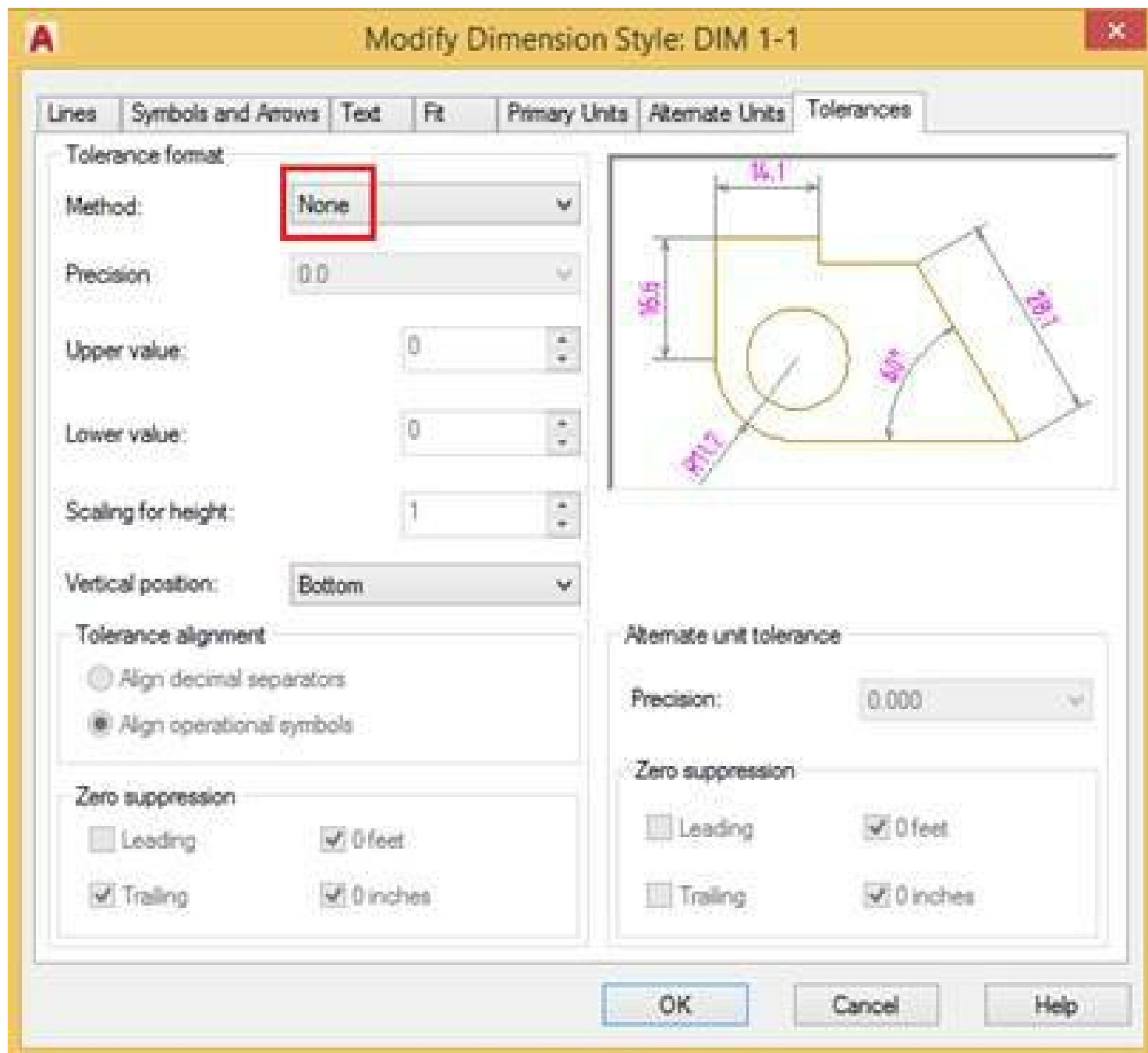
Zero suppression

☐ Leading ☒ Trailing

OK Cancel Help

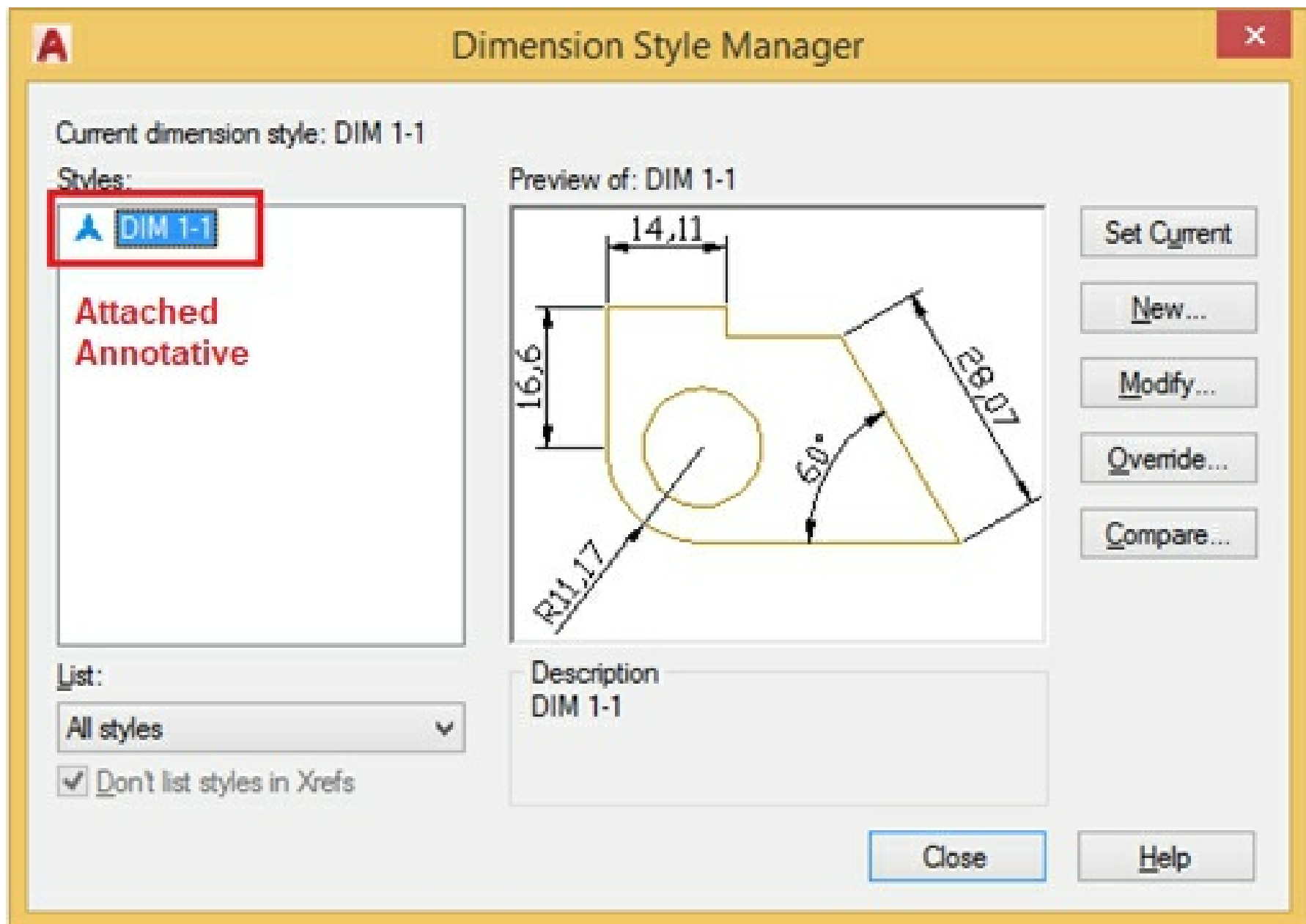


Alternate units tab keeps defaults

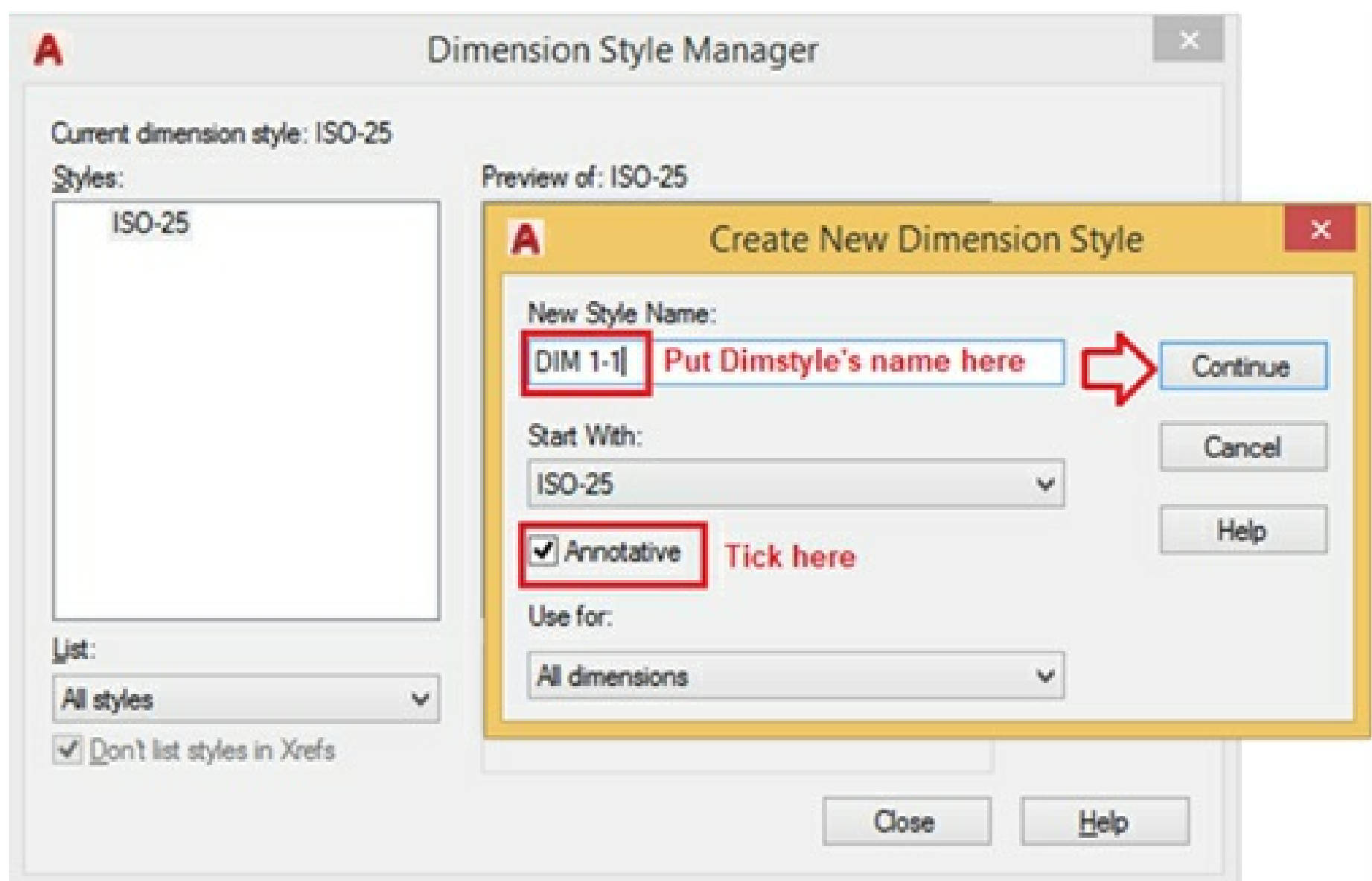


- Click OK to complete the setup.

The “DIM 1-1” style is Annotative attached, so it will have an Annotative star icon at the front.



In case you do not edit but create a new Dimstyle from the beginning. Instead of clicking Annotative in the Fit tab, you can always click Annotative at this initialization step.



After clicking **Continue**, you install as instructed above to set up DIM 1-1.

2. Annotation Scale

Annotative wants to promote its full strength, it must be combined with the Annotation Scale.

Have you memorized the 3 principles of drawing scale yet?

If not, then stop and read those 3 principles again. Always keep it in mind!

Principle 1: In Model space, the "display dimension" must always be noted with the "actual dimension" of the object, with a real draw scale by 1:1

Principle 2: In Model space, whatever the drawing scale, keep the original object and scale the title block

Principle 3: On the same paper size, all drawings, no matter what scale they are printed on, the basic height of the letters must be the same.

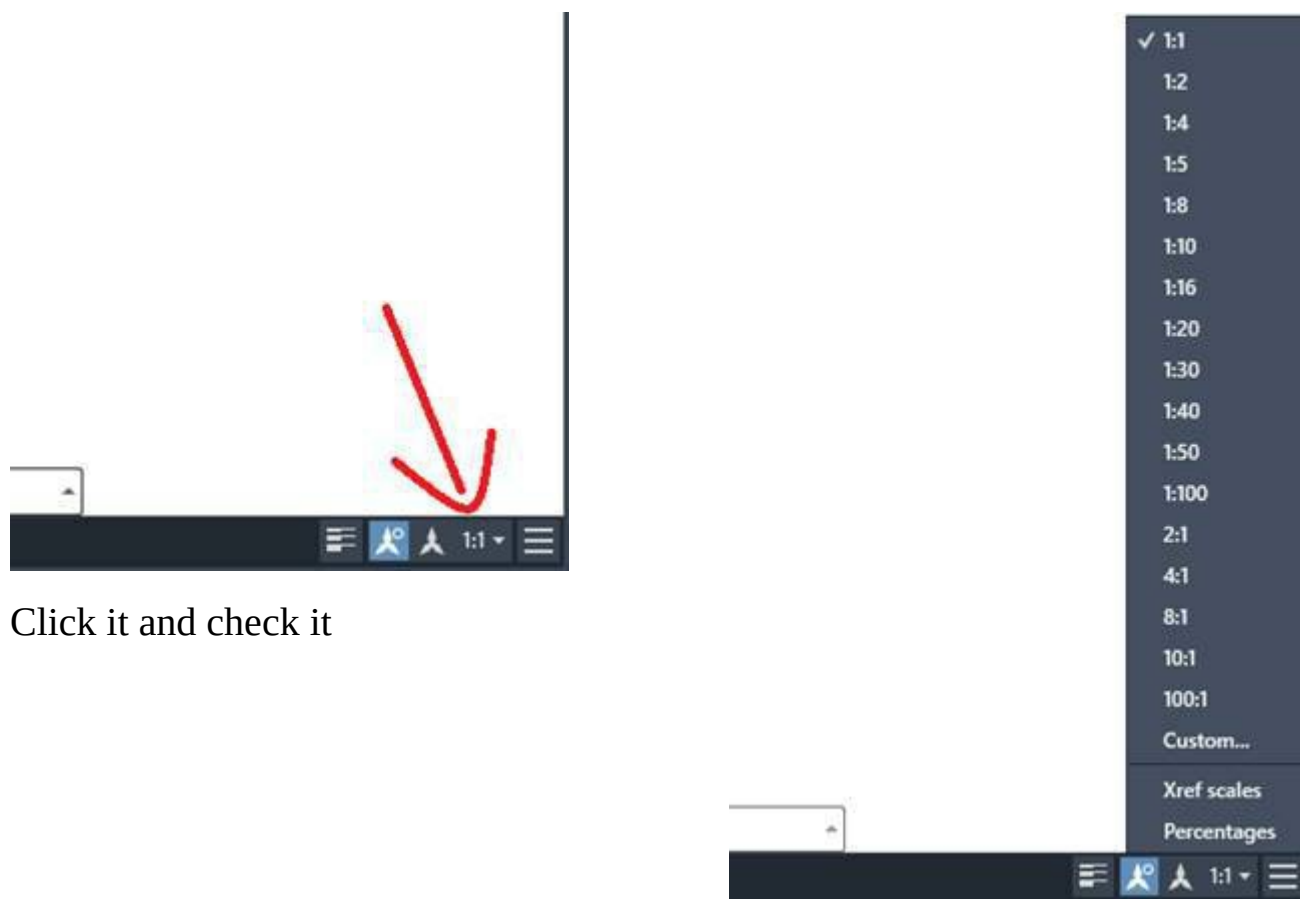
I have a straight line 100mm long. Requirements: regardless of the drawing scale, the size digits after printing on A4 paper must be 2.5mm high.

From principle 3, we deduce that:

- With a drawing scale of 1:1, the height of the dimension numerals on the Model is 2.5mm
- With a drawing scale of 1:2, the height of the dimension digits on the Model is $2 \times 2.5 = 5 \text{ mm}$
- With a 1:4 drawing scale, the height of the size digits on the Model is $4 \times 2.5 = 10\text{mm}$

In the old way, instead of you having to create 3 Dimstyles with Use overall scale off coefficients of 1, 2, 4 respectively, now you just need to create only 1 Dimstyle with the default Use overall scale off of 1. and each time you record the size with a certain drawing scale, you just need to choose the Annotation Scale coefficient corresponding to that drawing scale.

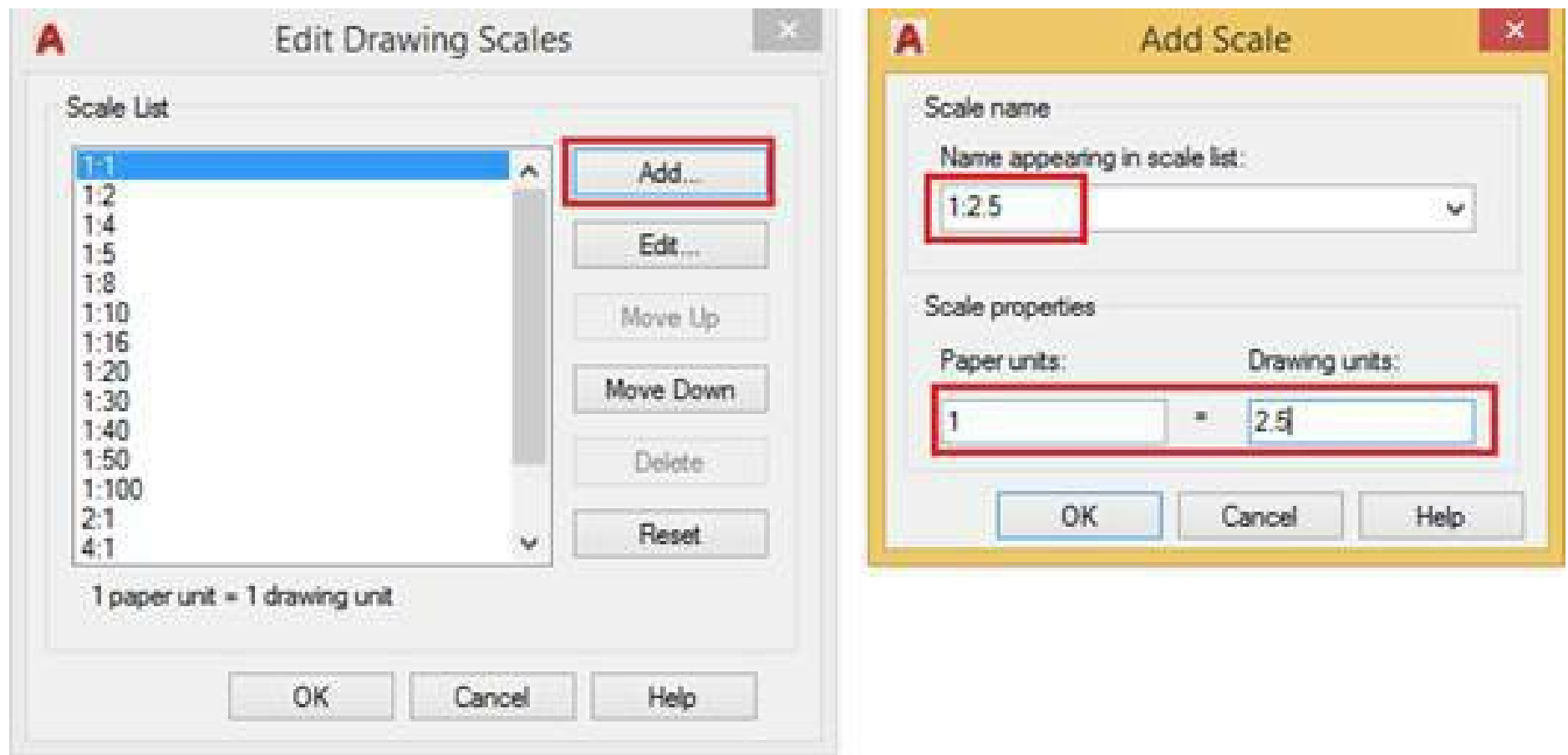
Do you see the box that says 1:1 in the bottom right corner of the screen?



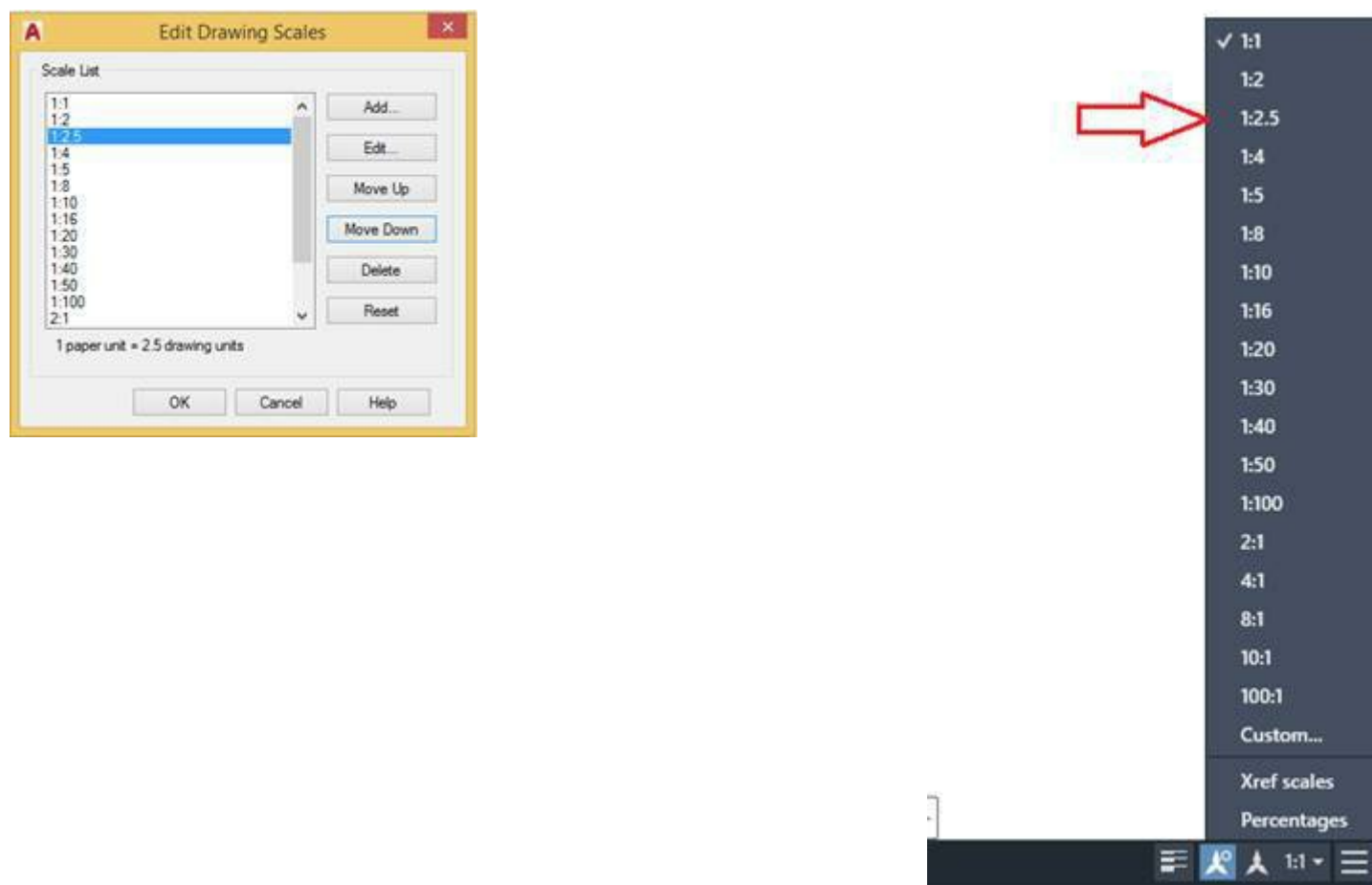
Click it and check it

You will see a series of Annotation Scales, which correspond to each drawing scale for which you plan to annotate the dimensions.

You can add another Annotation Scale, for example adding 1:2.5, by clicking Custom / selecting Add / entering a scale of 1:2.5 in the table and clicking OK.

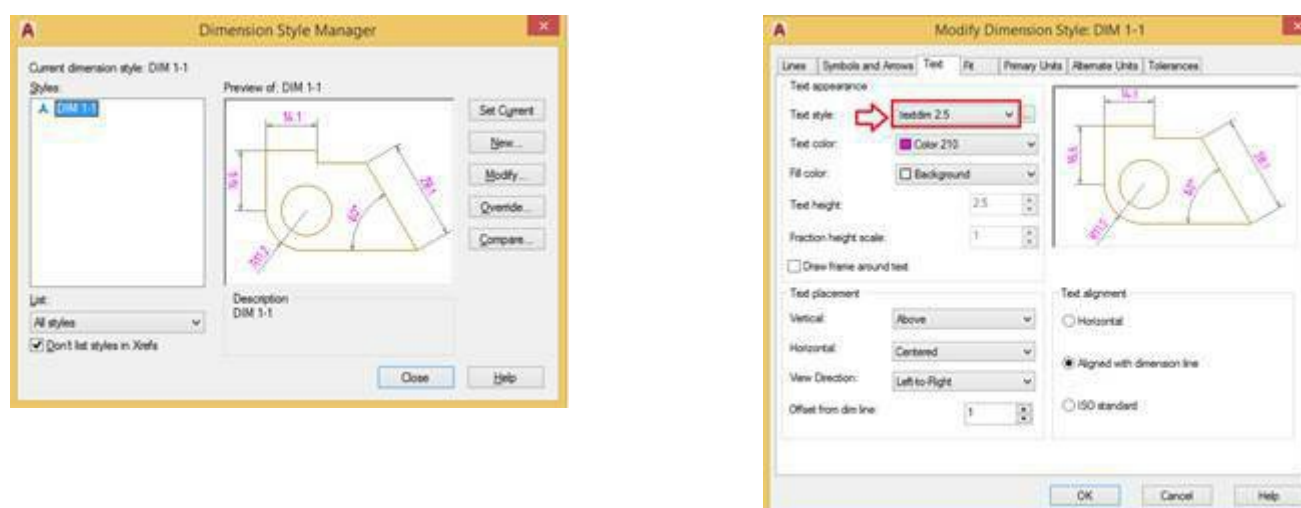


You see! 1:2.5 scale is added to the options list. You click Move Up, Move Down to move it up and down and get the result as shown on the right.



3. Dimensioning method for multiple drawing scales using only a single Dimstyle

I have equipped you with the only Dimstyle "DIM 1-1" with Annotative, with the font used textdim 2.5 (with font iso.shx) for A4 size paper.

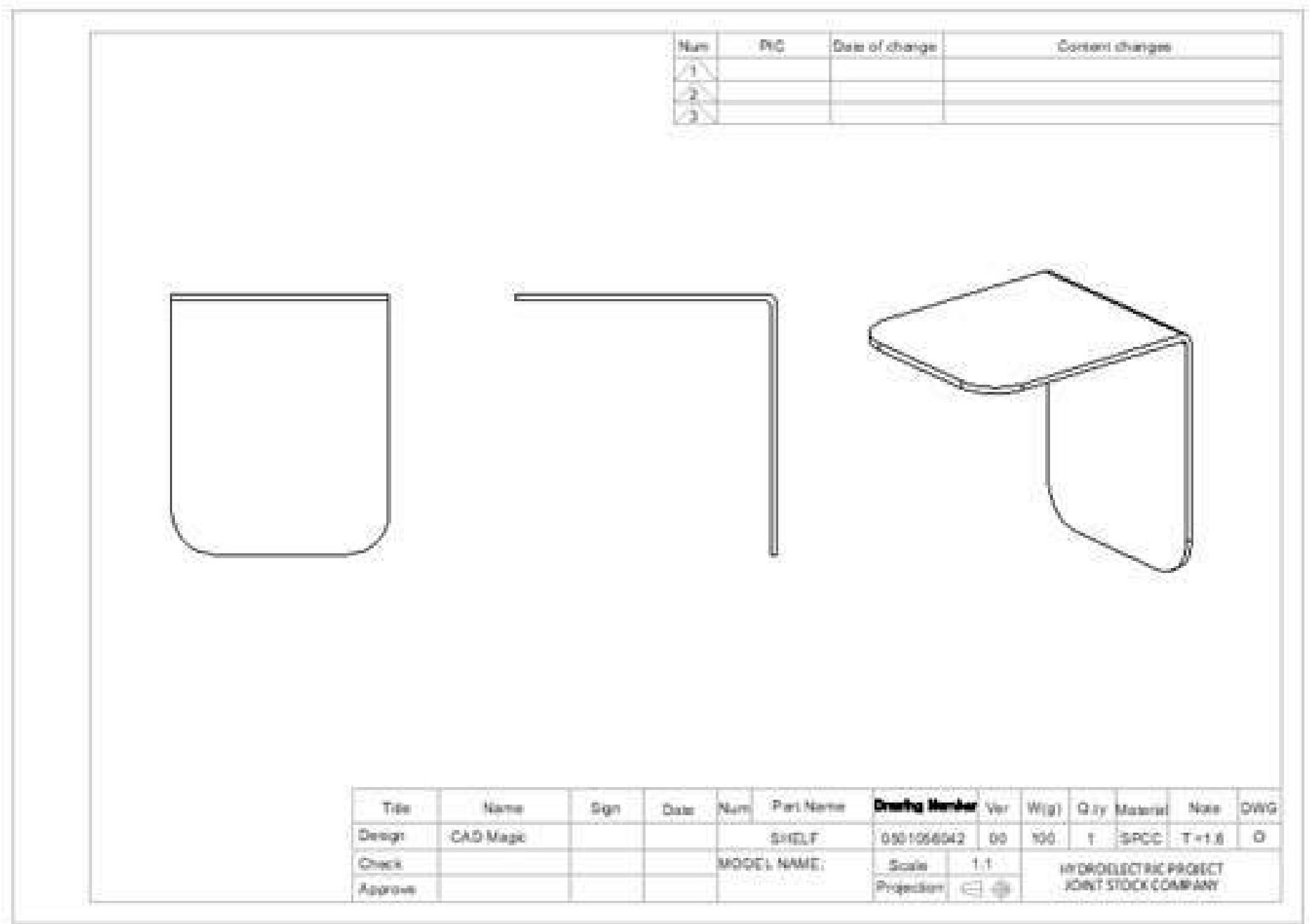


To annotate the dimensions for multiple drawing scales and use only a single Dimstyle, do the following:

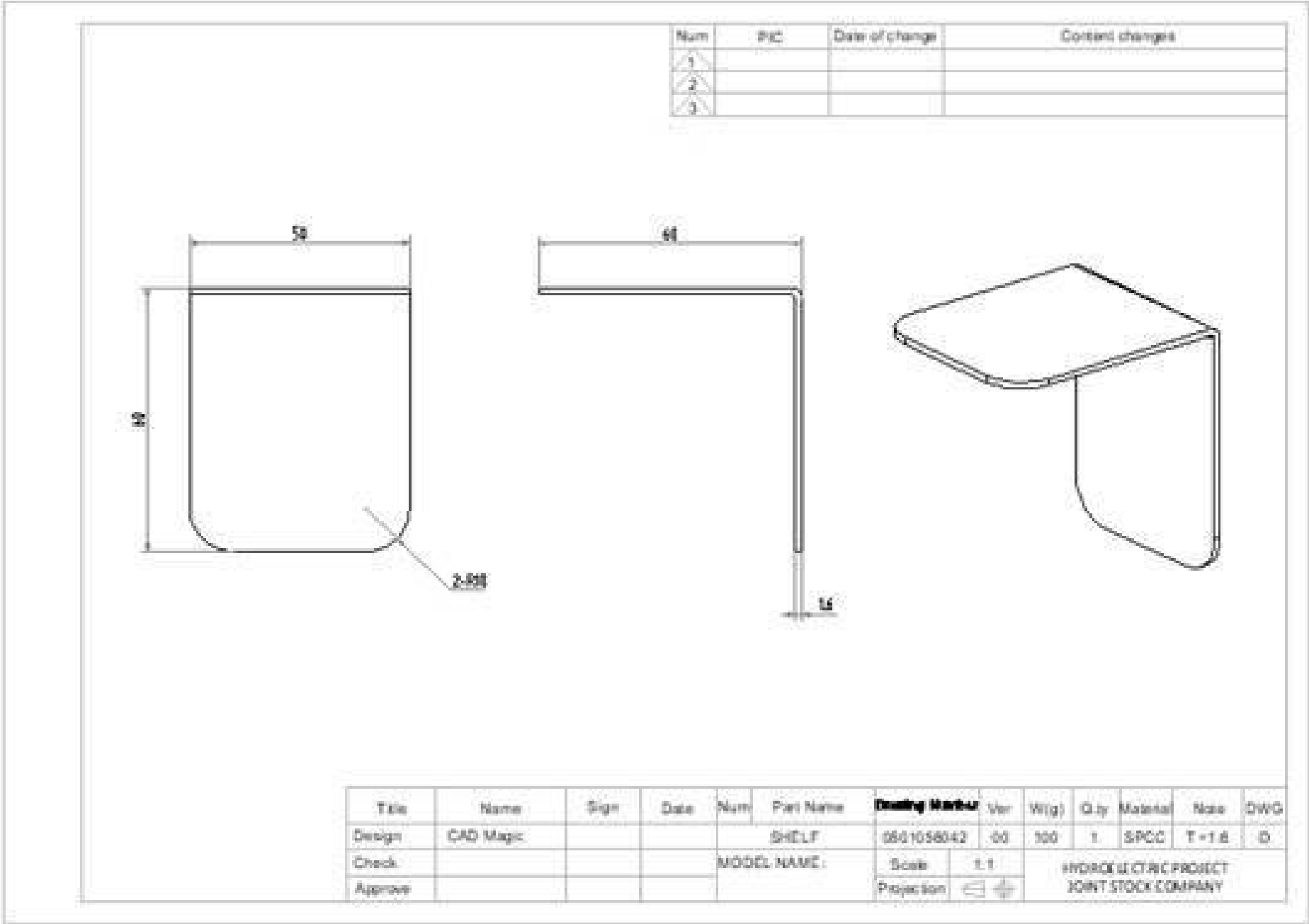
The principle is: "**With how much drawing scale, choose the same Annotation Scale coefficient to proceed with dimensioning**"

Example 1: Note the dimension for the drawing scale 1:1

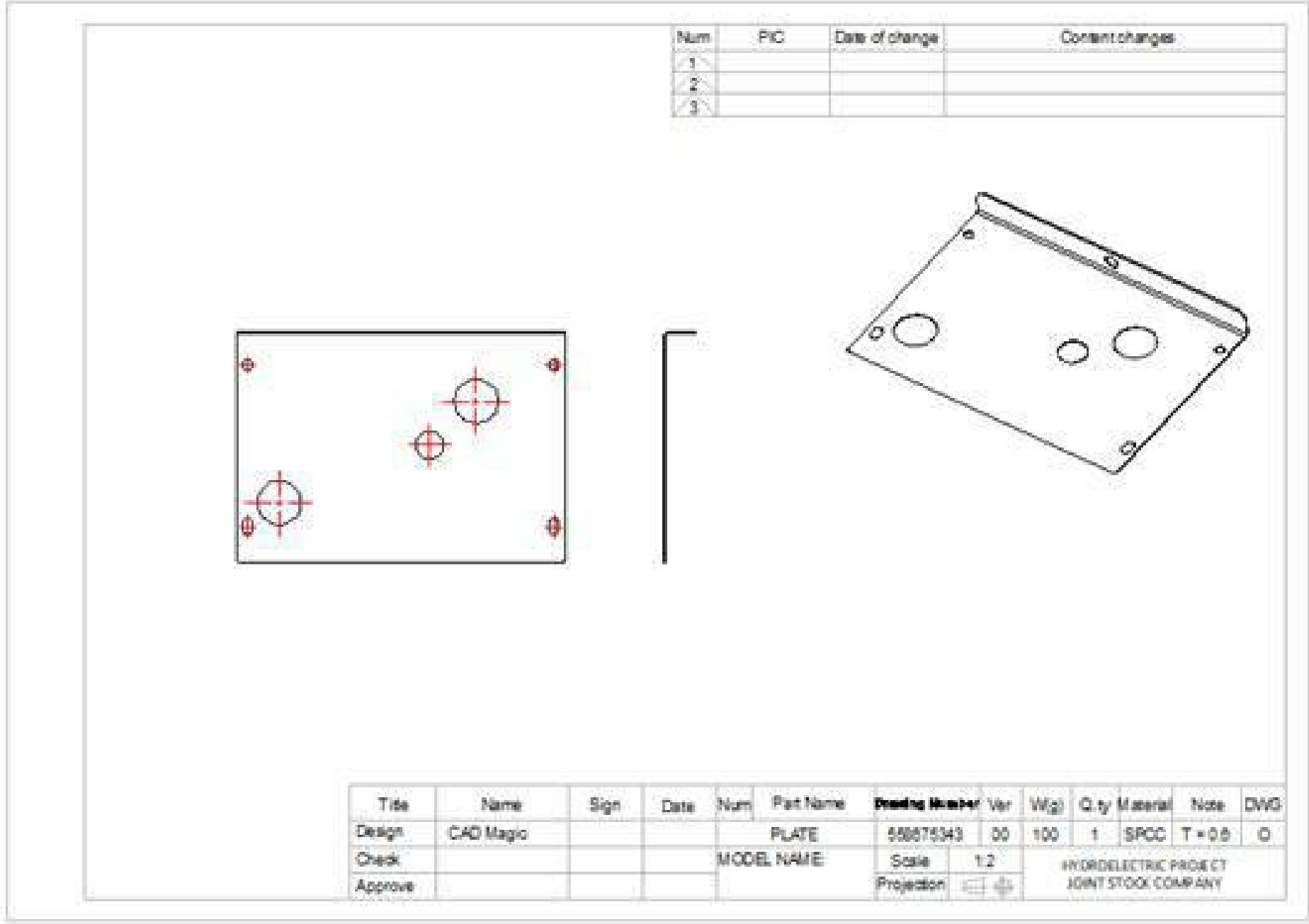
I have a stamped part, after exporting the projection and placing it on A4 paper, it has a 1:1 drawing scale on AutoCAD like this.



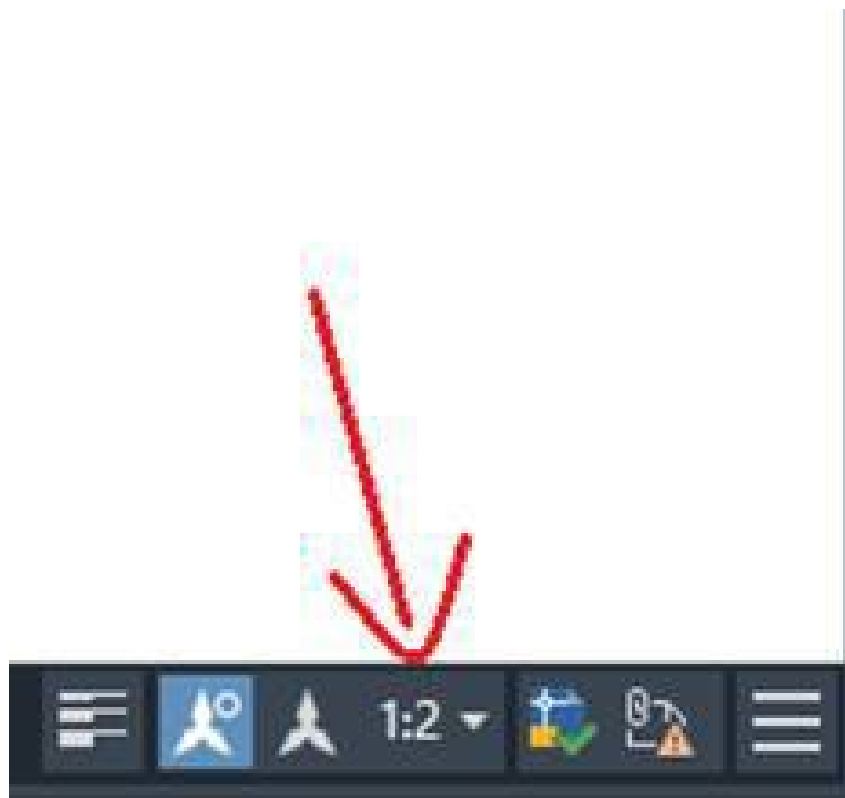
- Select “DIM 1-1” style as current dimstyle
- Before starting to note the size, you click and the lower right corner select the Annotation Scale ratio of 1:1 (default is 1:1 already)
- Here, you use the basic commands to note the dimension as usual.



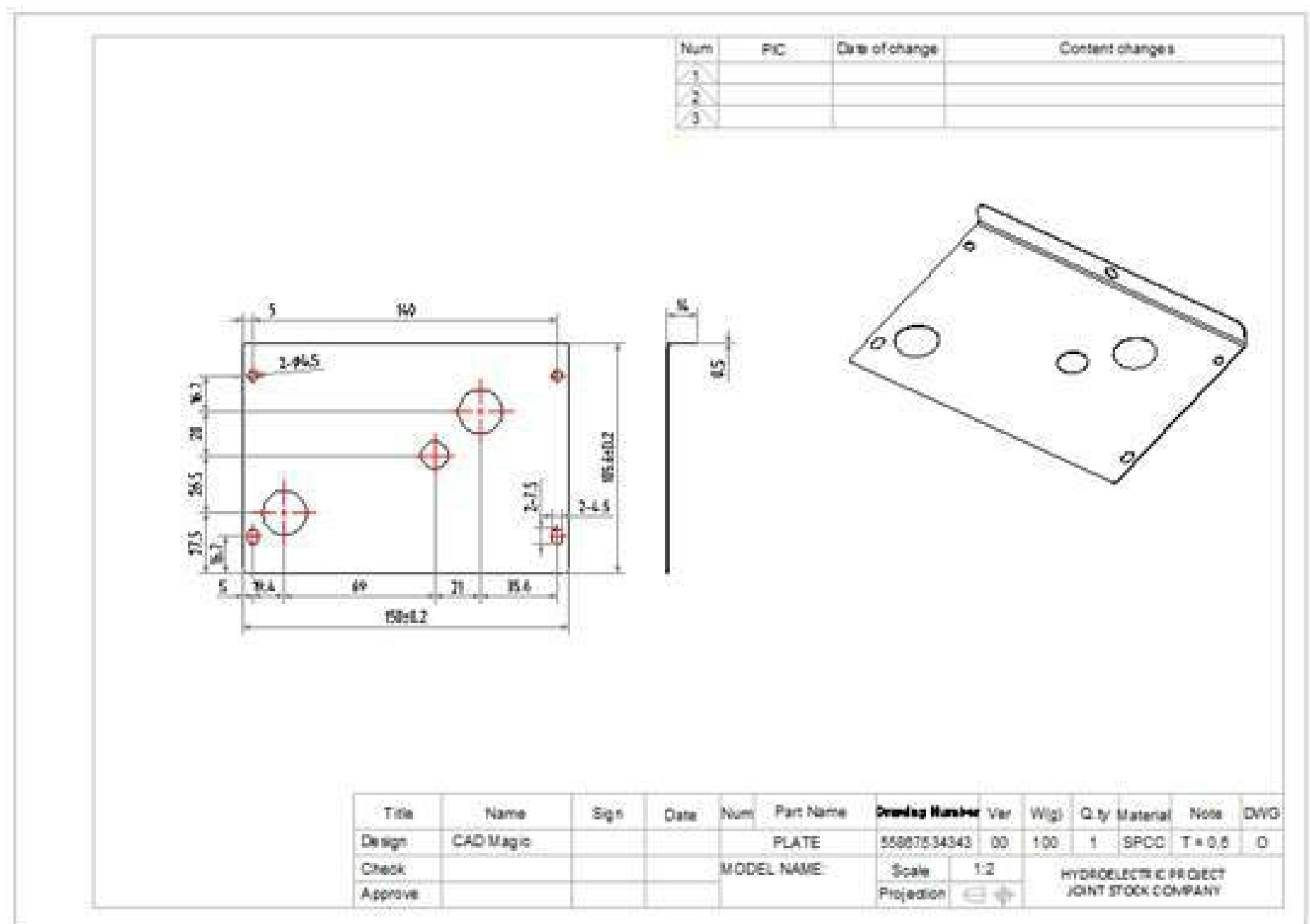
Example 2: Similar, but in this case, note the dimension for the drawing scale by 1:2 - The projection is exported to AutoCAD and placed in a "doubly-magnified title block"



- Click on the bottom right corner of the screen, select Annotation Scale by 1:2

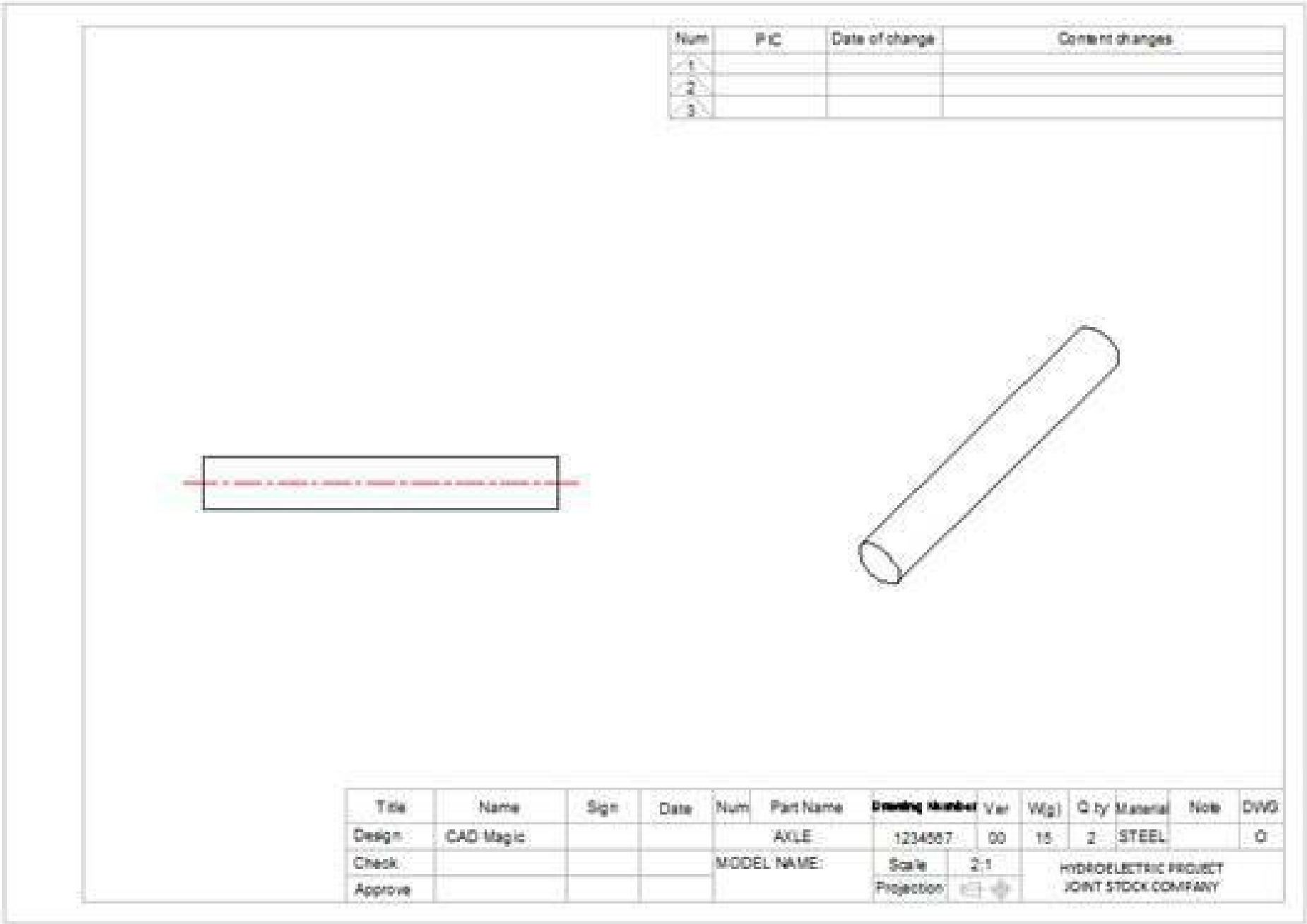


- Note dimensions as usual

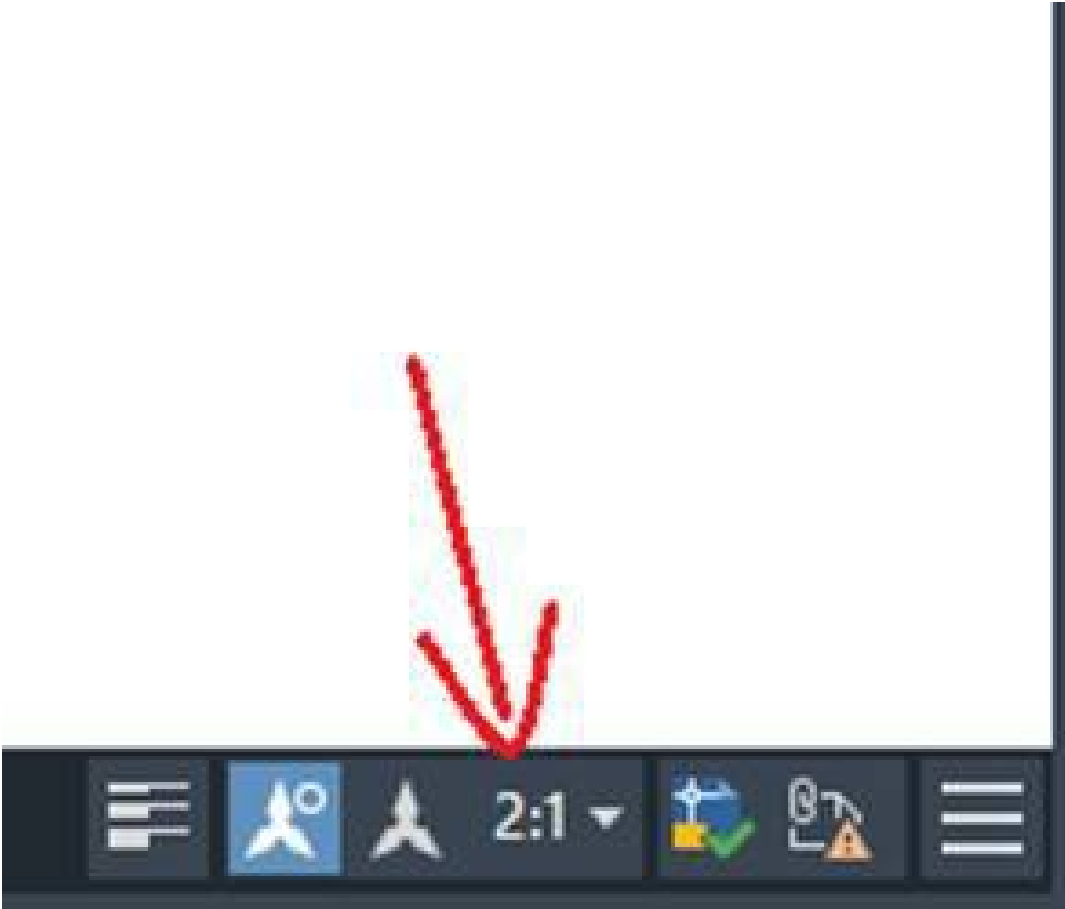


Example 3: Still the same, but this time the drawing scale is 2:1

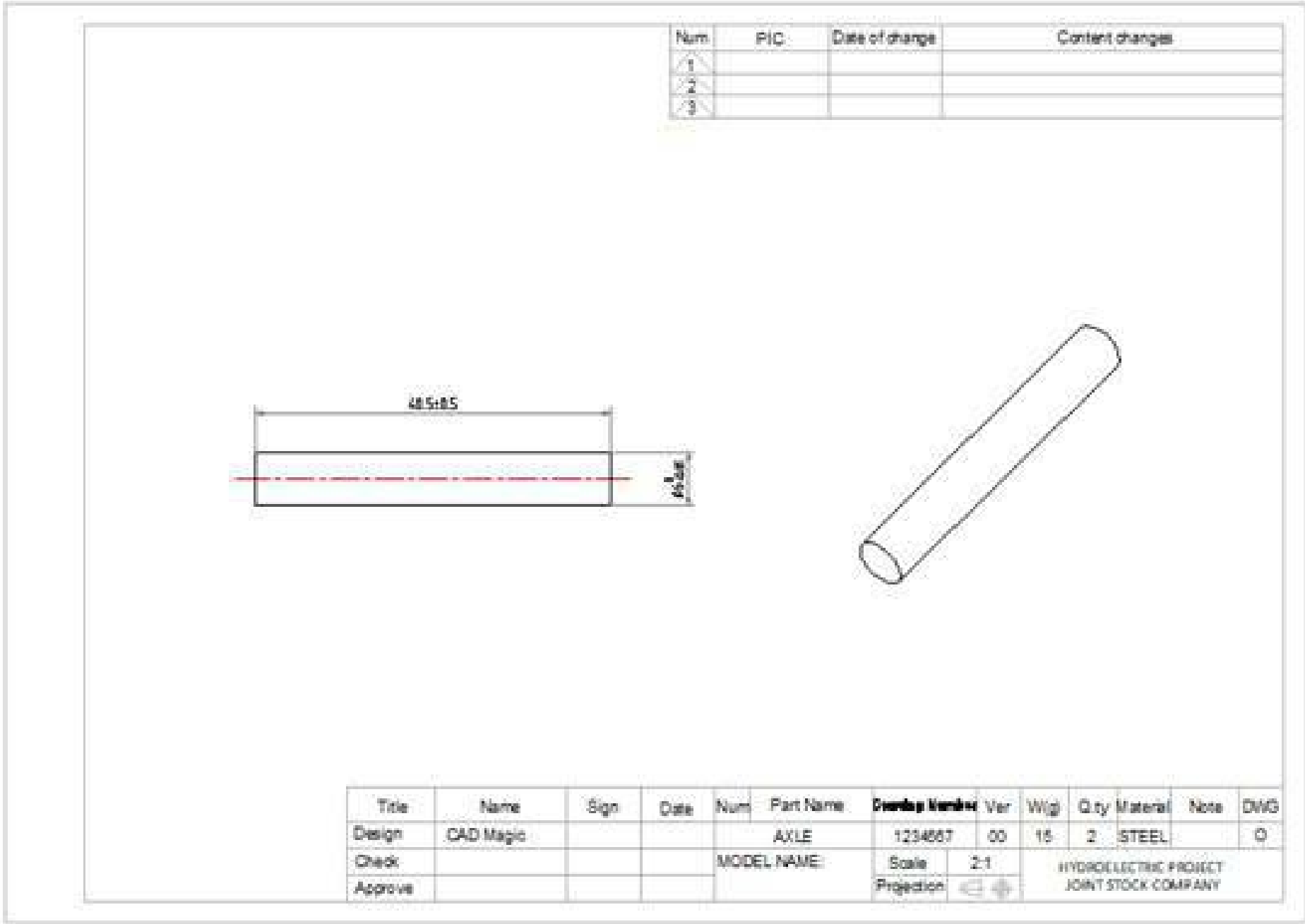
- The projections are exported to AutoCAD and placed in a "title block that has been scaled by 0.5 times"



- Click on the bottom right corner of the screen, select Annotation Scale by 2:1



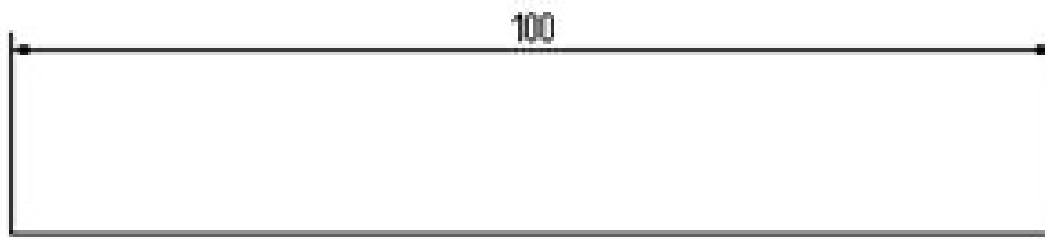
- Note dimensions as usual



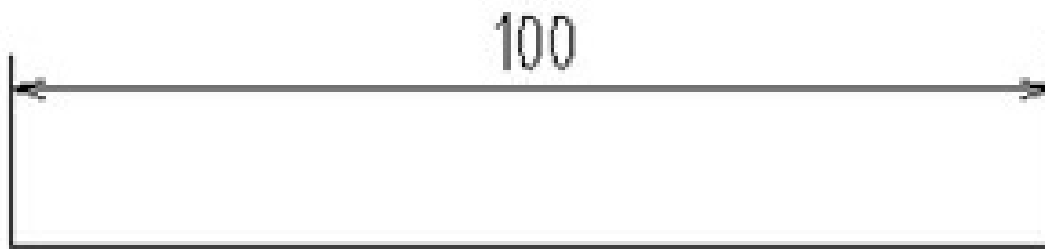
Thus, it is clear that I have only used a single Dimstyle "DIM 1-1" to annotate the dimensions for all the drawing scales just by changing the Annotation Scale, corresponding to each drawing scale

Do the same for all other ratios you come across.

By changing the Annotation Scale, the size of the dimension line and the dimension value is increased proportionally. Therefore, after printing to paper, make sure the font size and dimension lines are always in sync.



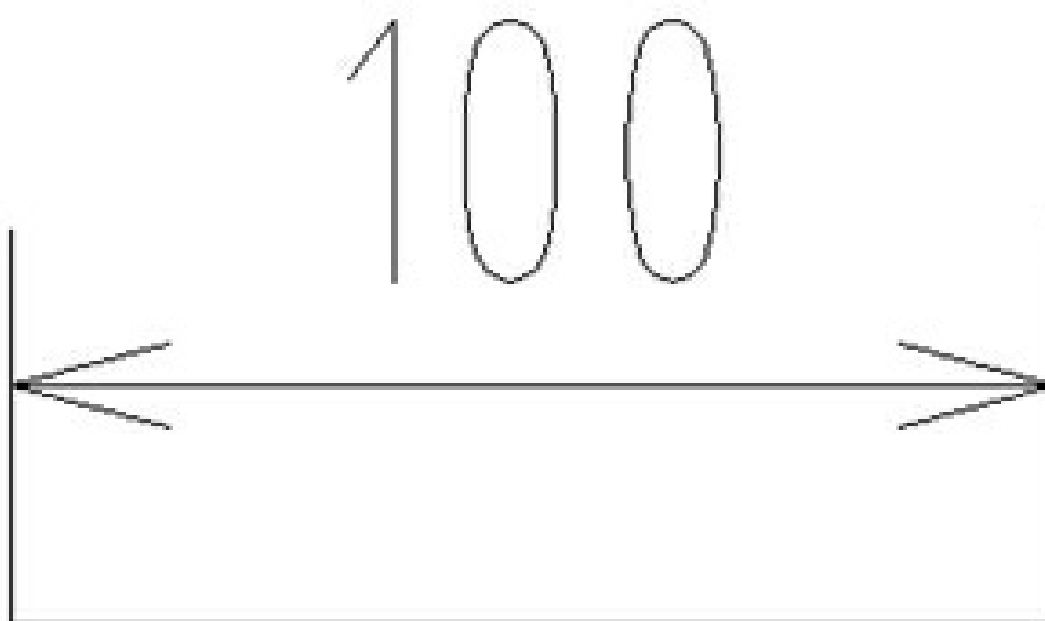
Drawing scale = 1:1



Drawing scale = 1:2



Drawing scale = 1:5



Drawing scale = 1:10

Summary:

With normal drawings (without detail area) with a "real draw scale" of 1:1, its dimensioning method is a combination of the Annotative and Annotation Scale.

Principle: Use only 1 Dimstyle with Annotative attached. With any drawing scale, choose the corresponding Annotation Scale coefficient and make dimension notes for the object in that title block

Chapter 14: Method For Noting The Dimension Of The Detail Area

"Detail area" often appears when we are in the process of noting dimension. At any position, it is necessary to note the dimension, but with the current drawing scale, that position is too small, it cannot see after printing out. Therefore, we need to use "Detail area" at that position to enlarge that area to note the dimension. At that time, the reader can see the structure and dimension of that place.

For the "detail area" drawing, because it has been enlarged, the real draw scale is different from 1:1. At this time, on a drawing, people need to manage both types of scale, the "drawing scale" and the "real draw scale". The method of noting dimension for "detail area" is a combination of Scale Factor, Annotative, and Annotation Scale.

What the hell is a scale factor?

Do you remember the formula (*) and (**):

$$\text{Scale factor} = \frac{1}{\text{Real Draw Scale}} = \frac{\mathbf{D}}{\mathbf{A}} = \frac{\text{Display}}{\text{Actual}} \quad (***)$$

The "scale factor" determines what "real draw scale" you will draw to?

Example: Real draw scale = 2:1 = 2, $\Rightarrow$ *Scale factor* = $1/2 = 0.5$

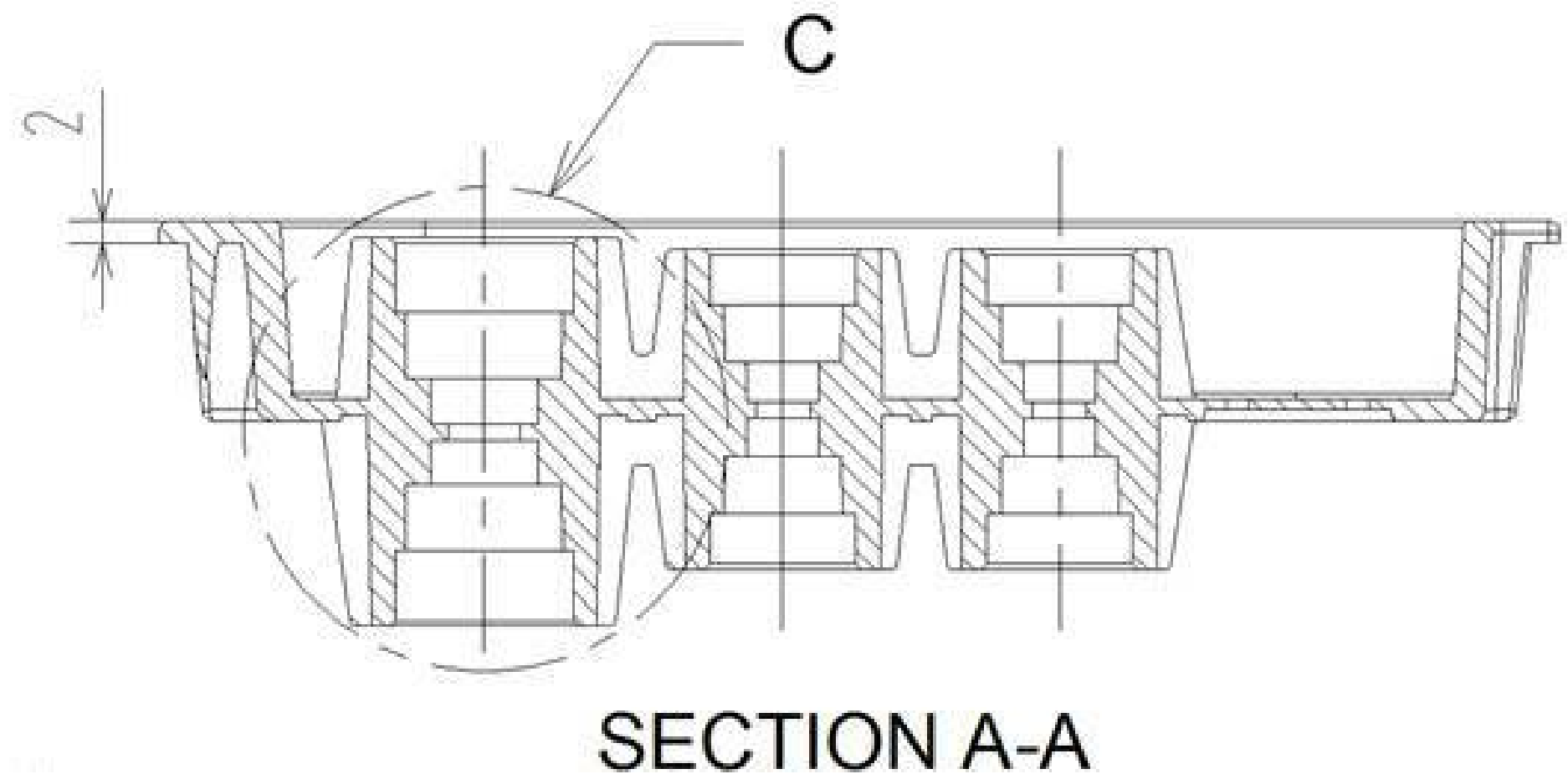
*The **scale factor** is the hidden person behind, controlling the displayed value on the screen (which is what we "desire") when noting the size. Therefore, we need to manage it, control it to work for us.*

1. Method of creating a detail area in AutoCAD

Detail area drawing can be exported from 3D software or created by yourself in AutoCAD.

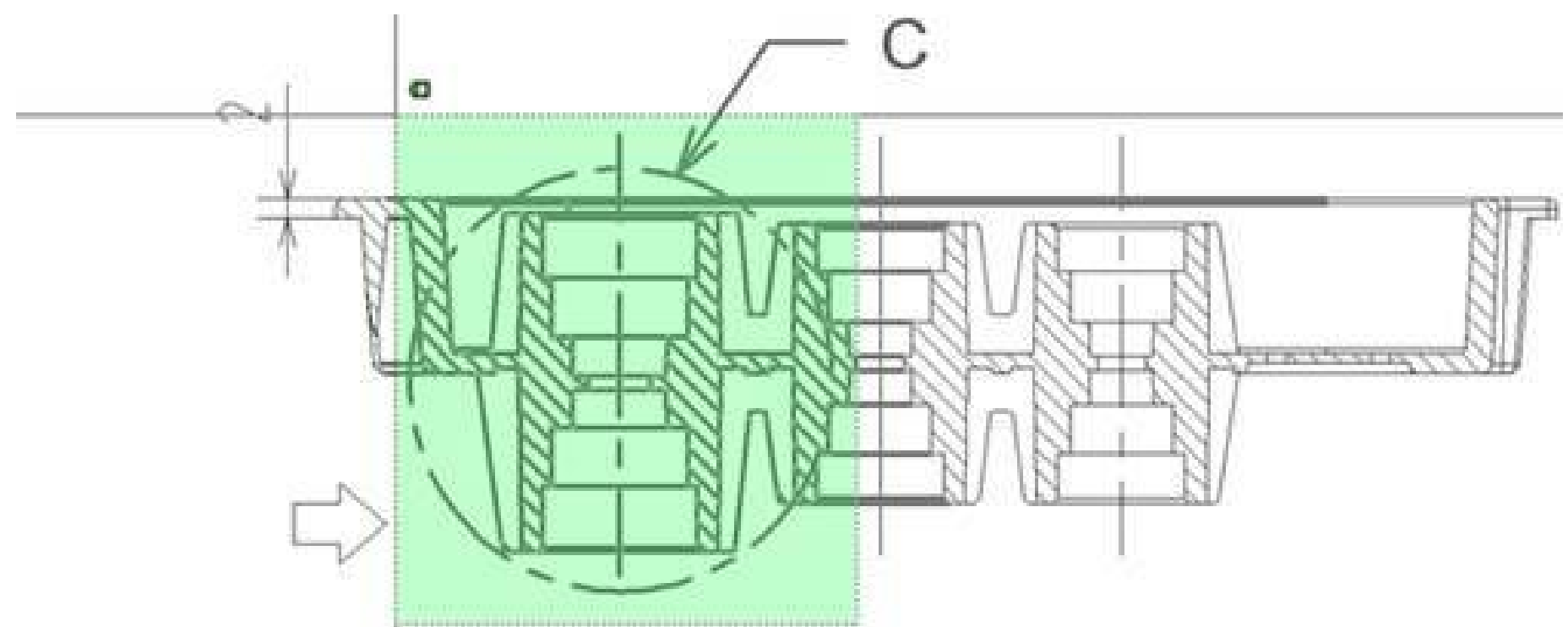
To create an extract, we only need to use 2 main commands, the Extrim command, and the Scale command. The way to do it is simple, I call this the manual way.

For example, I have a cross-section of a jig (a waterworks part) with a drawing scale of 1:2, I need to magnify the 3/8" quick-connect bead cavity area for easy dimensioning.

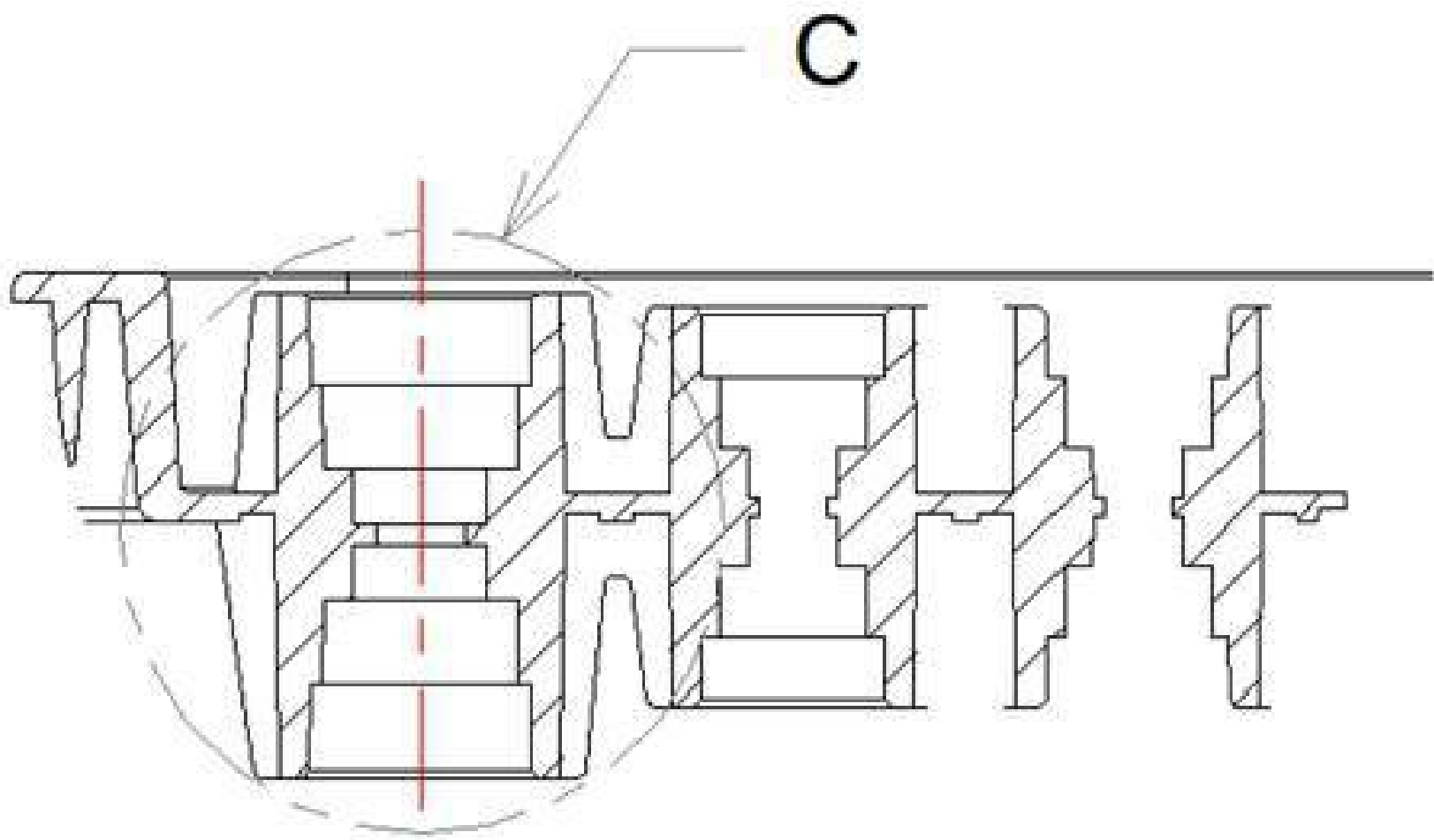


Step 1: Copy the entire C area to another place:

- Type CO command to Copy
- Sweep the mouse from right to left (Crossing Window method) to make sure all objects surrounding the circle are selected.

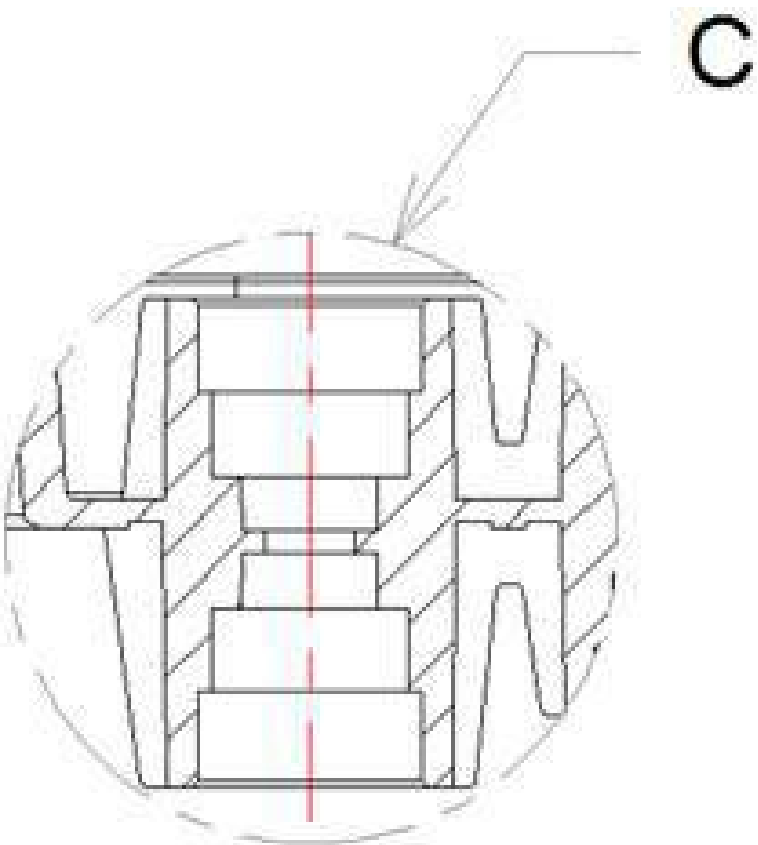


I got results like this



Step 2: Trim the excess outside the contour C

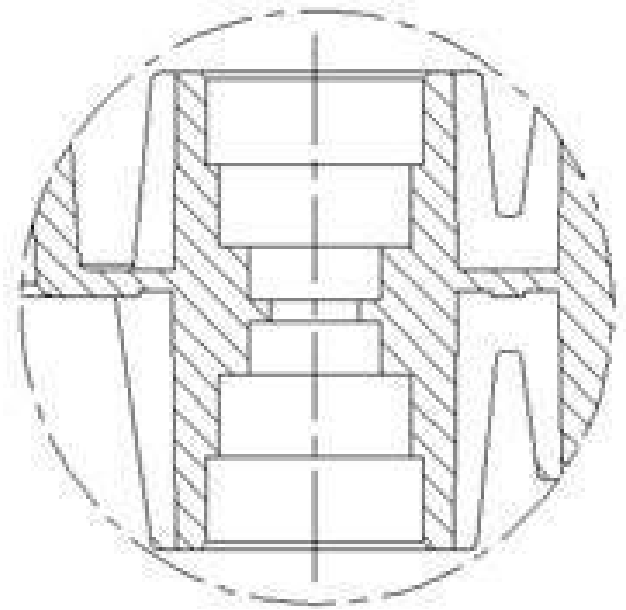
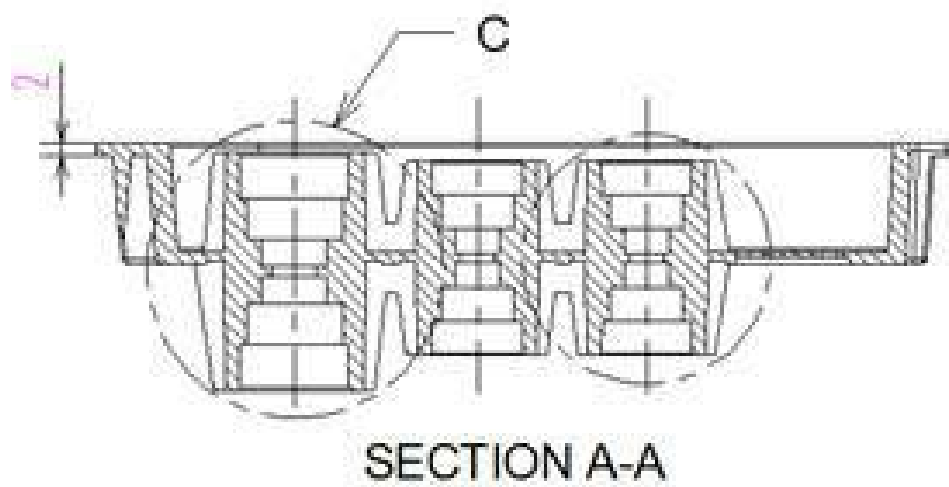
- First I use Explode to break the hatch
- Use the Extrim command to cut the outside of the C contour
- Type Extrim/ Click on the C circle/ Click on the area outside the C contour
- Erase the remaining parts outside the C
- The result is C with “real draw scale” by 1:1



Step 3: Enlarge the extracted image to a certain scale to suit the drawing layout (preferably coefficients 2; 2.5; 4; 5; 10, ...)

Here, I magnify C 2 times

Step 4: Make note of the extract according to the correct “**real draw scale**”.



Step 5: Note the dimension for Detail area

Invite readers to continue reading part 2.

2. The method of noting the dimension of the detail area

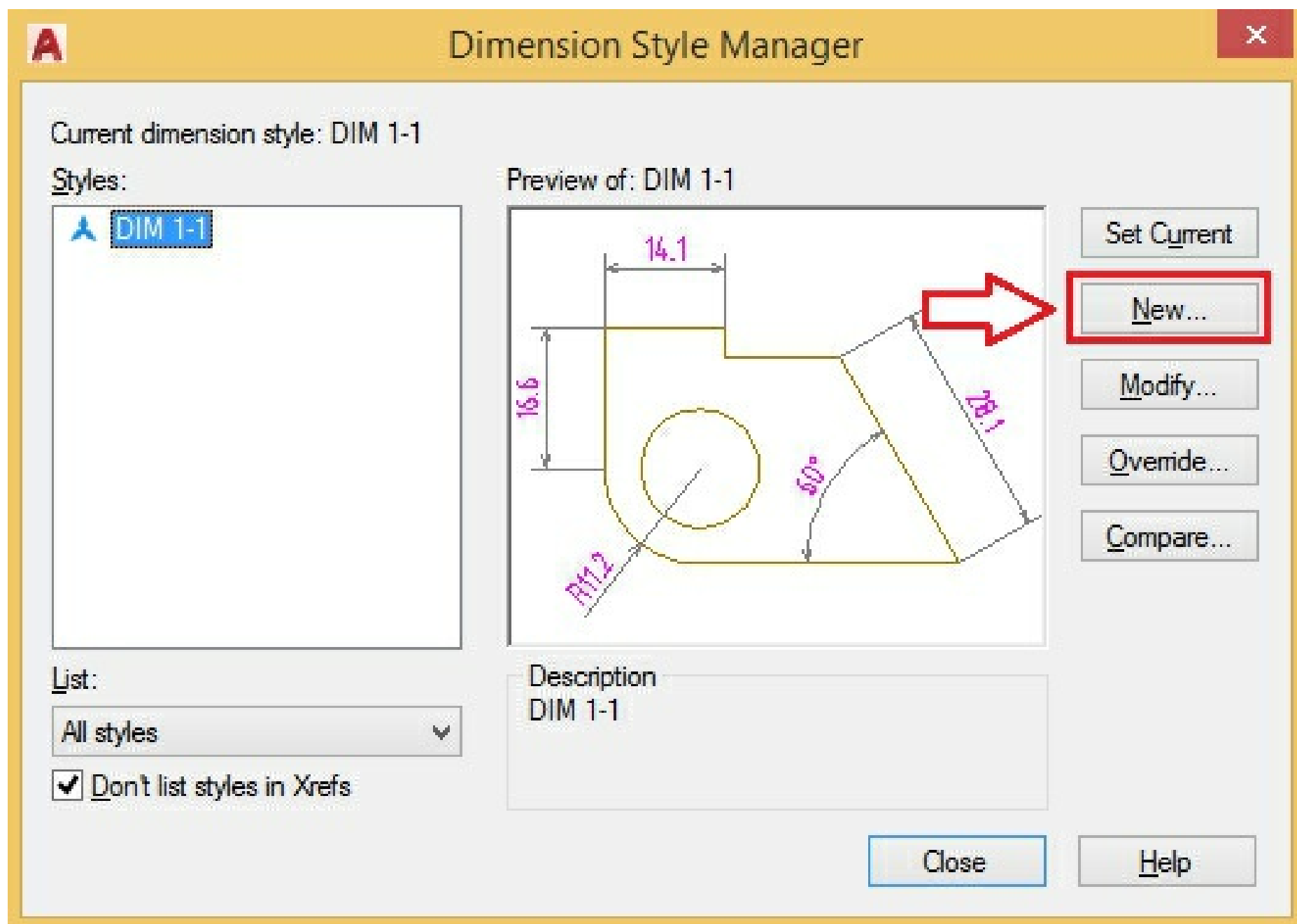
Principle:

For each "real draw scale" of the extracted areas, we create a corresponding Dimstyle with Annotative attached and adjust the Scale factor accordingly. With any drawing scale, choose the Dimstyle corresponding to that drawing scale, and at what drawing scale the drawing is, choose the Annotation Scale coefficient corresponding to that drawing scale and then proceed to note dimension for detail area.

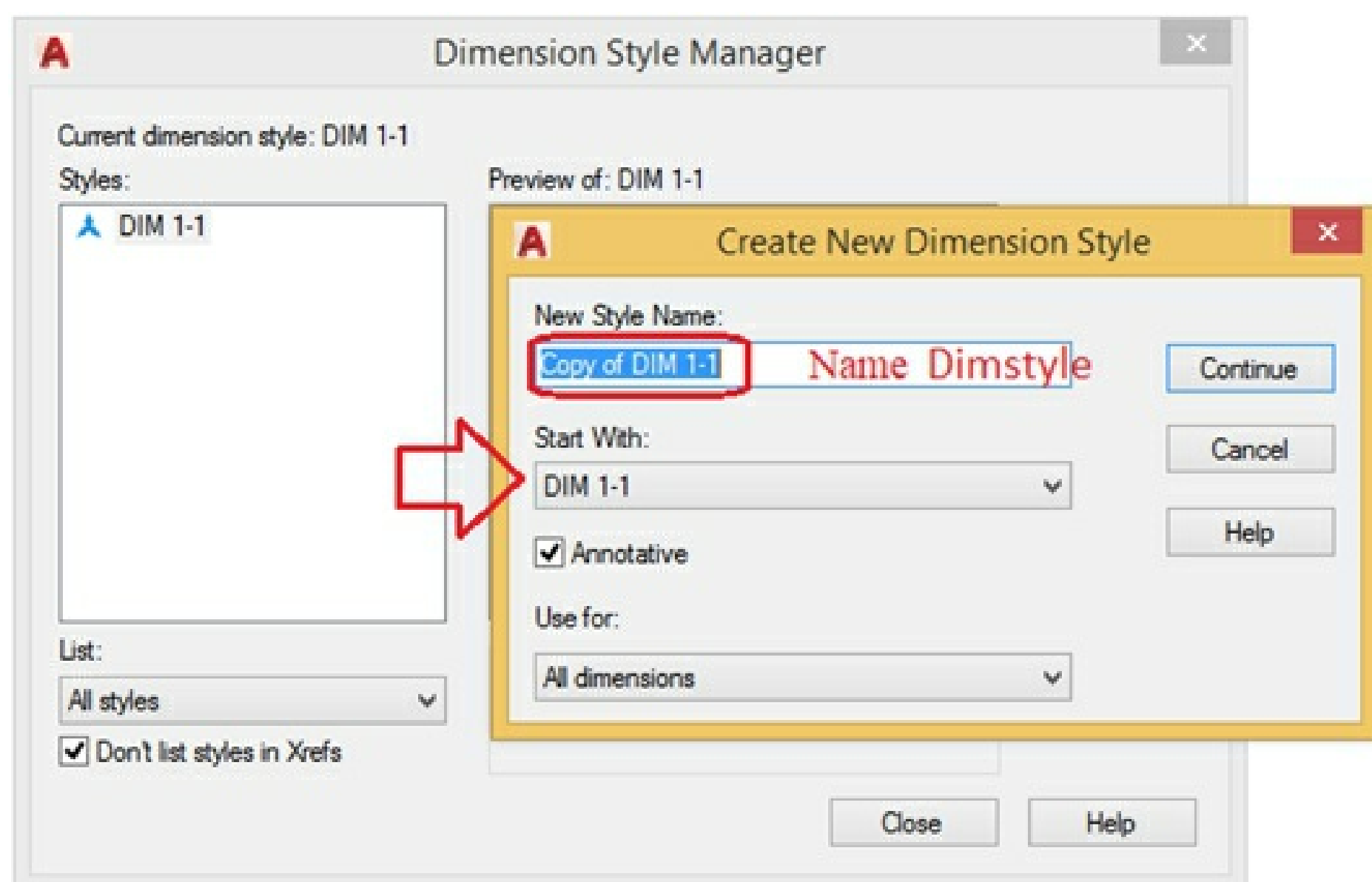
2.1. How to set up Dimstyle to note the dimension of the detail area drawing.

Dimstyle for detail area can be easily created in the fastest way is to copy almost all properties from "**DIM 1-1**".

- Type the command: **DD** to enable the Dimstyle dashboard
- Click New to create a new Dimstyle



- Name the new Dimstyle and let **Start With** be “DIM 1-1”



Assuming the detail area has a real draw scale of 2:1, we set up a Dimstyle named “**DIM 2-1**”

- Click *Continue*

Create New Dimension Style

New Style Name:

Start With:

☒ Annotative

Use for:

- Just change the **Scale factor** in the **Primary Units** tab. Set Scale factor = 0.5
- Other information remains unchanged

New Dimension Style: DIM 2-1

Lines Symbols and Arrows Text Fit **Primary Units** Alternate Units Tolerances

Linear dimensions

Unit format:

Precision:

Fraction format:

Decimal separator:

Round off:

Prefix:

Suffix:

Measurement scale

Scale factor:

Zero suppression

☐ Leading ☒ Trailing

Sub-units factor: ☐ 0 feet ☐ 0 inches

Sub-unit suffix:

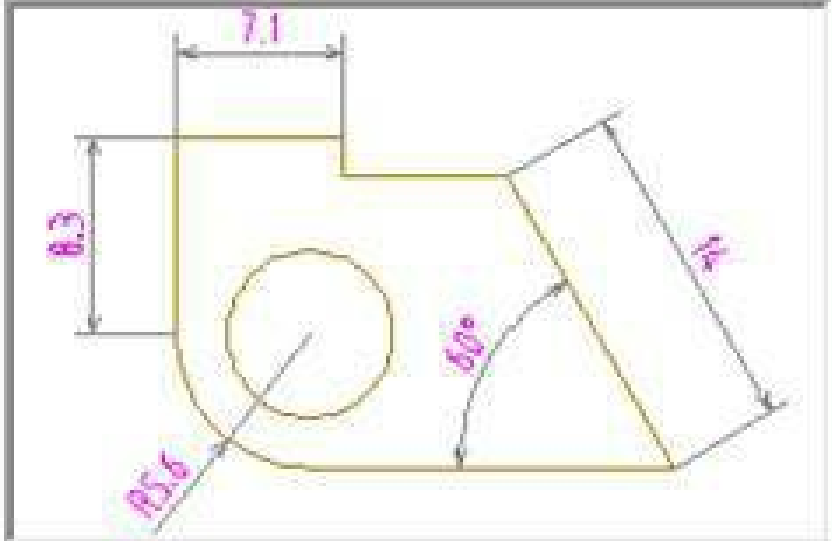
Angular dimensions

Units format:

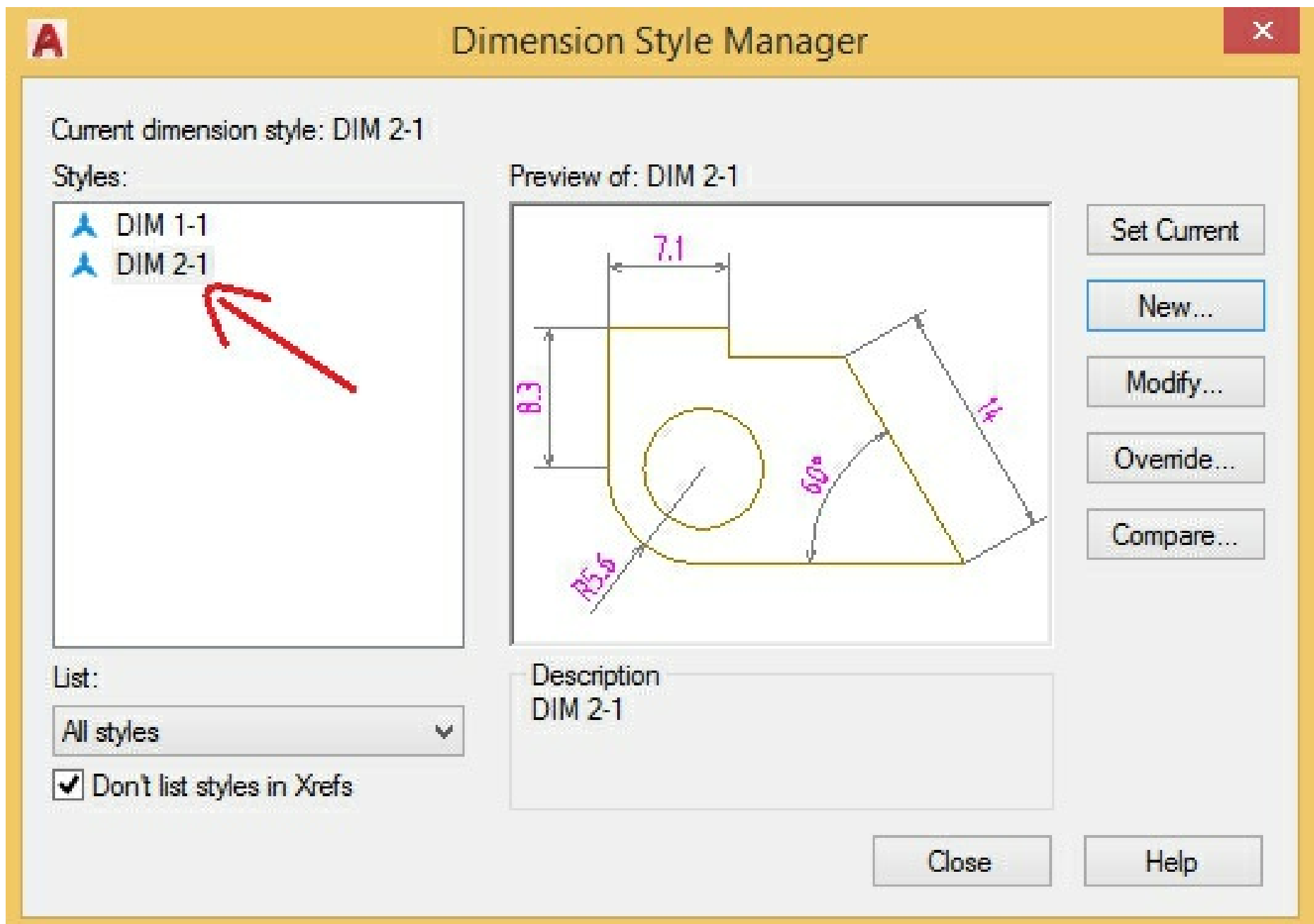
Precision:

Zero suppression

☐ Leading ☒ Trailing



And you have a new Dimstyle, DIM 2-1 is copied all the properties from DIM 1-1, the only difference is the Scale factor.



Do the same with another real draw scale of another detail area (if any), enter the Scale factor according to the formula (***)

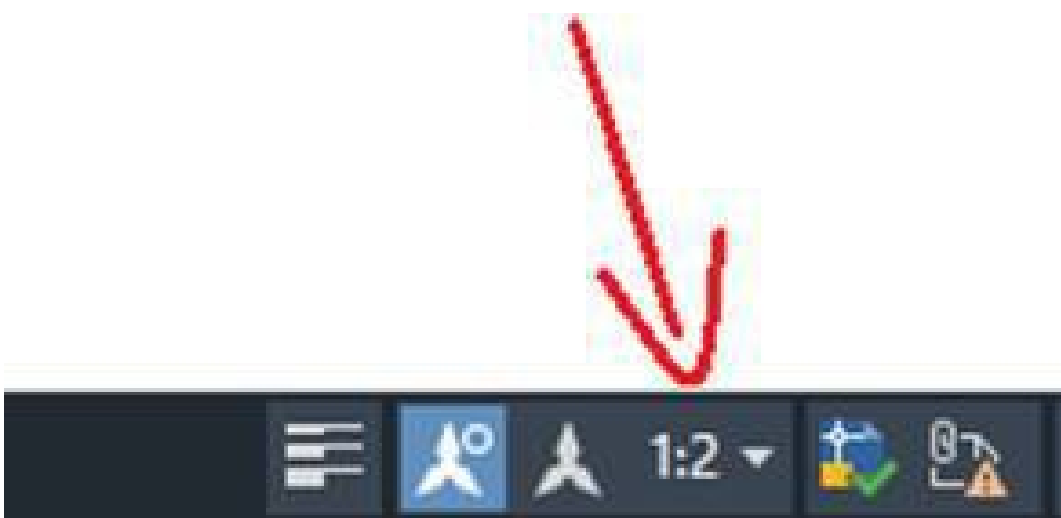
Specifically:

- If the detail area has a real draw scale of 5:1, then Scale factor = 0.2
- If the detail area has a real draw scale of :1, then Scale factor = 0.25

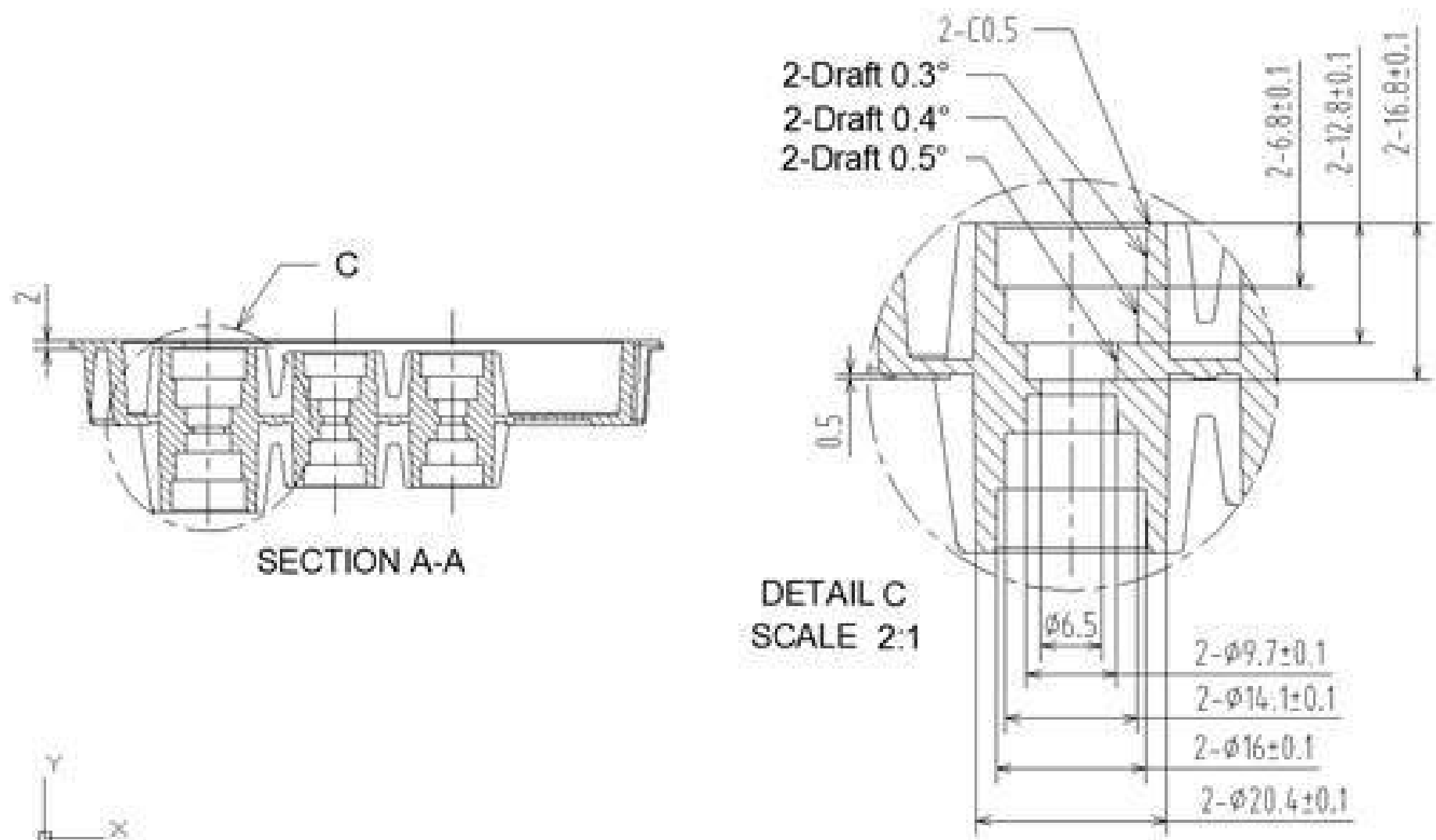
2.2. Note dimension for detail area

Example: Continue to note the dimensions for the C

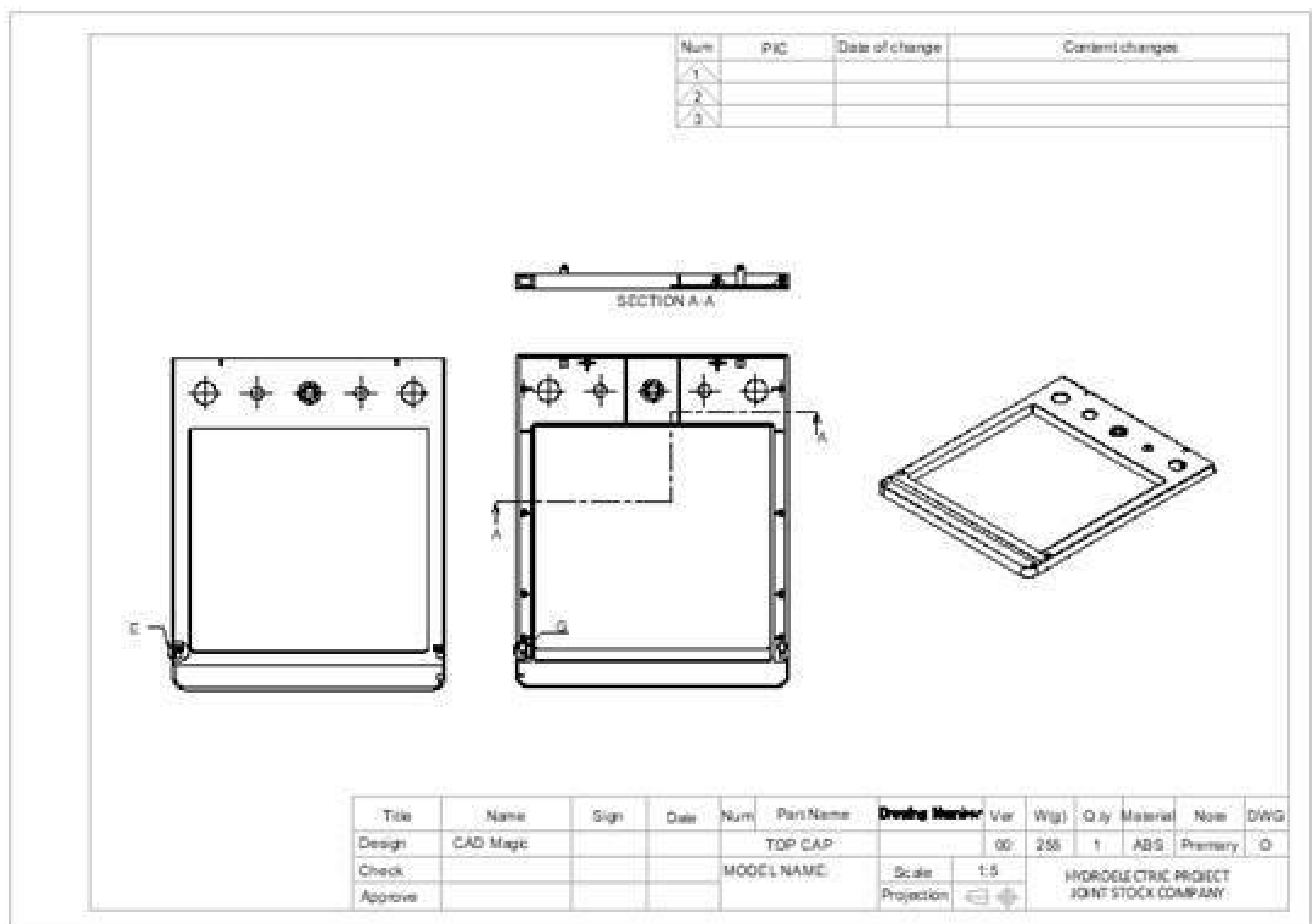
- Because the real draw scale is 2:1, we set "DIM 2-1" as current.
- Choose Annotation Scale = 1:2 because drawing scale is 1:2



- Dimension note:



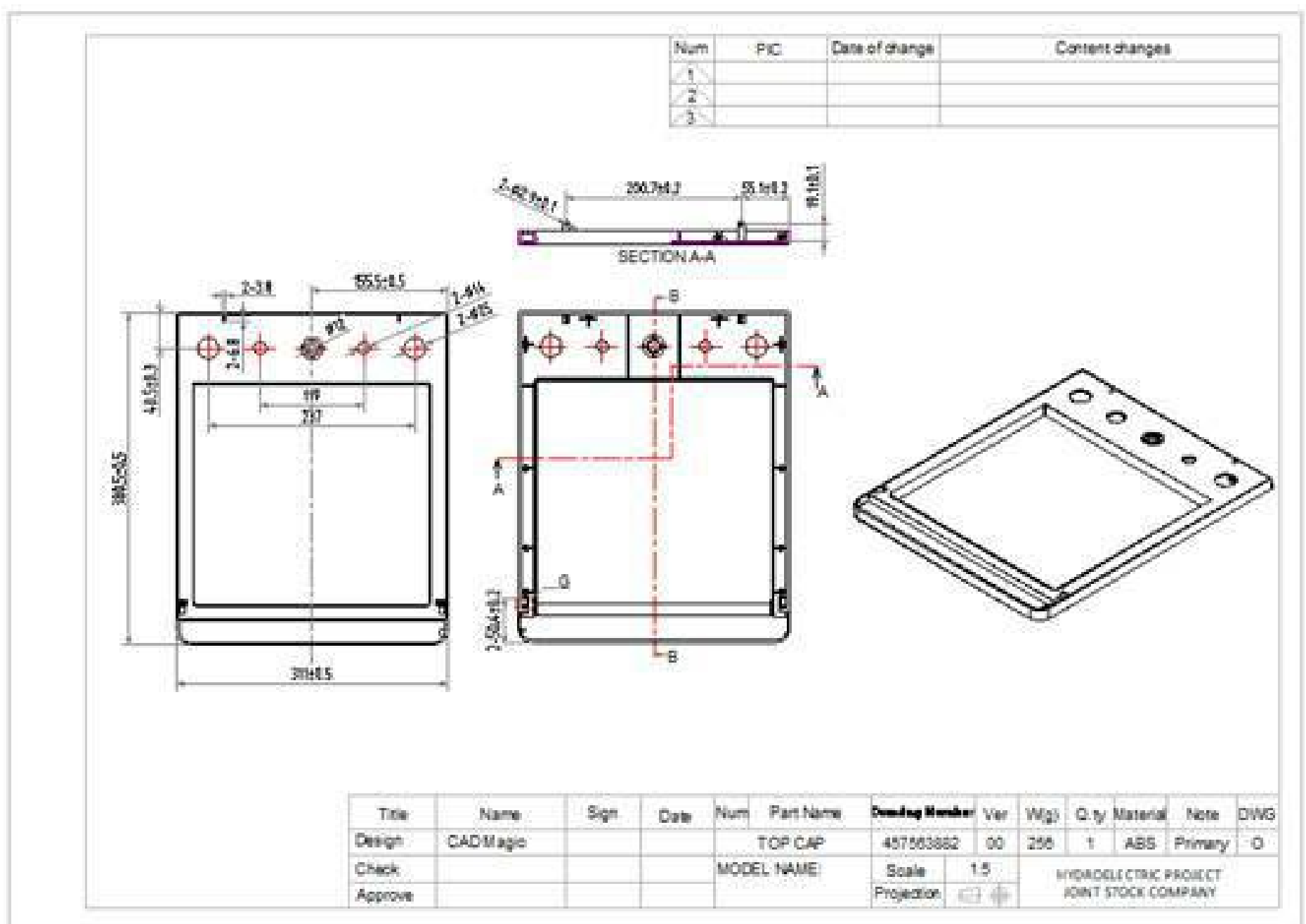
Example 2: Suppose we have a drawing with a drawing scale of 1:5.



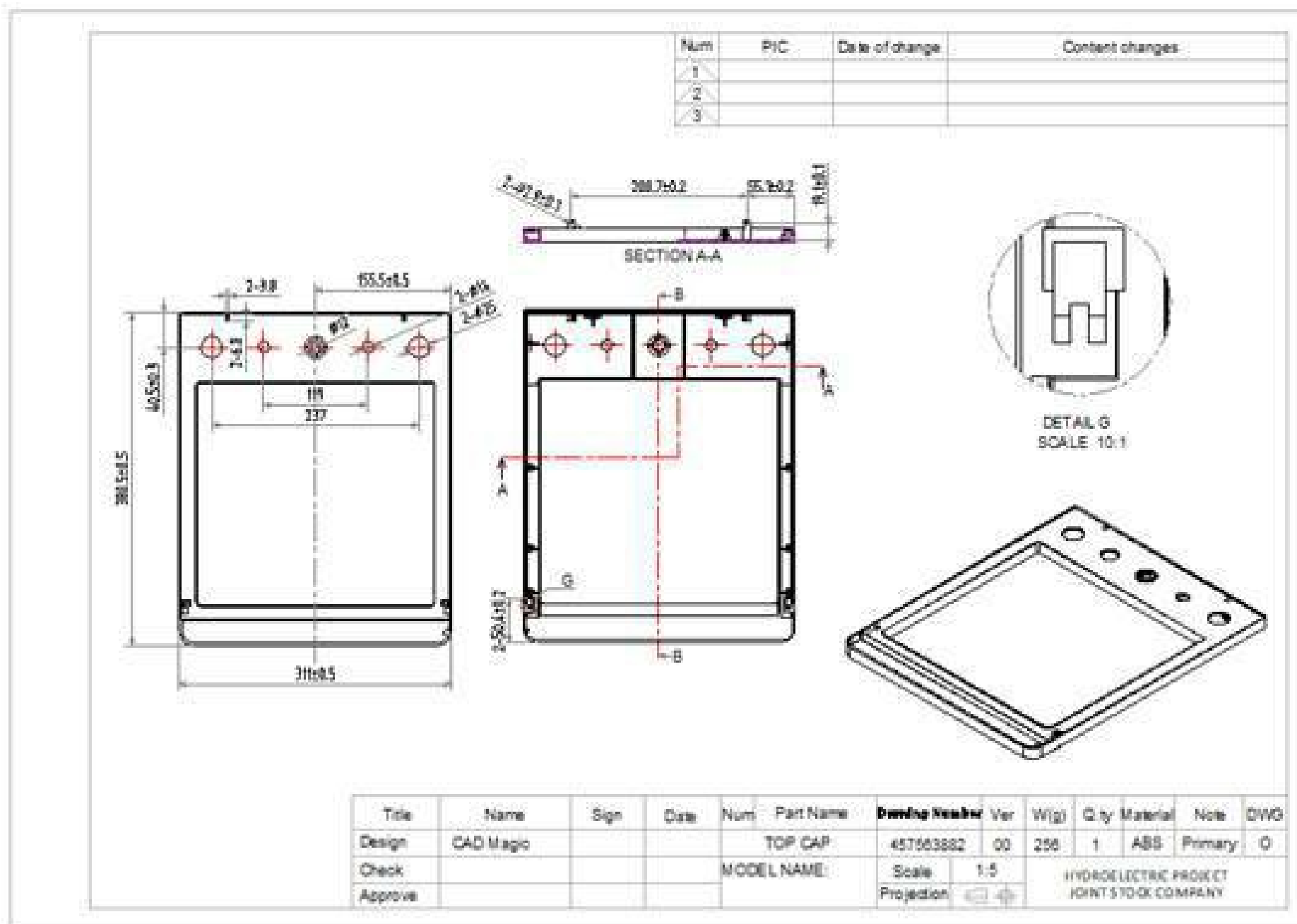
- Open Dimstyle panel, double click on “DIM 1-1” style to set it as active Dimstyle
- Select Annotation Scale = 1:5 in the lower right corner of the screen



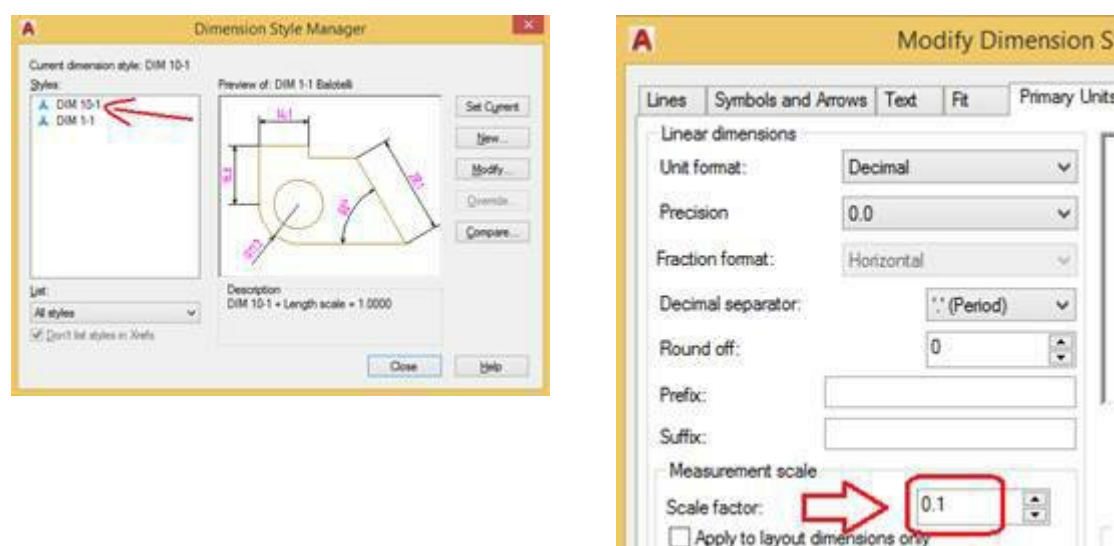
- Carry out dimensioning (and add tolerances for important dimensional positions to be controlled)



- In the process of noting the size, realizing that an area needs to be clarified, I noted as D. With the drawing layout, D needs to zoom in 10 times to see clearly. I annotated the extract as DETAIL D – SCALE 10:1 so that the drawing reader knows that area D is being enlarged 10 times.



Due to the real draw scale of 10:1, we need to create a new Dimstyle called “DIM 10-1”, copy from “DIM 1-1” and have a Scale factor = $1/10 = 0.1$



- Double click on “**DIM 10-1**” to set it as the current Dimstyle (if it is already existing then ignore it)
- Still have to keep the Annotation Scale = 1:5 (because the drawing has a drawing scale = 1:5)
- Specify the dimensions for the extract D by "**DIM 10-1**"

(drawing for illustrative purposes only)

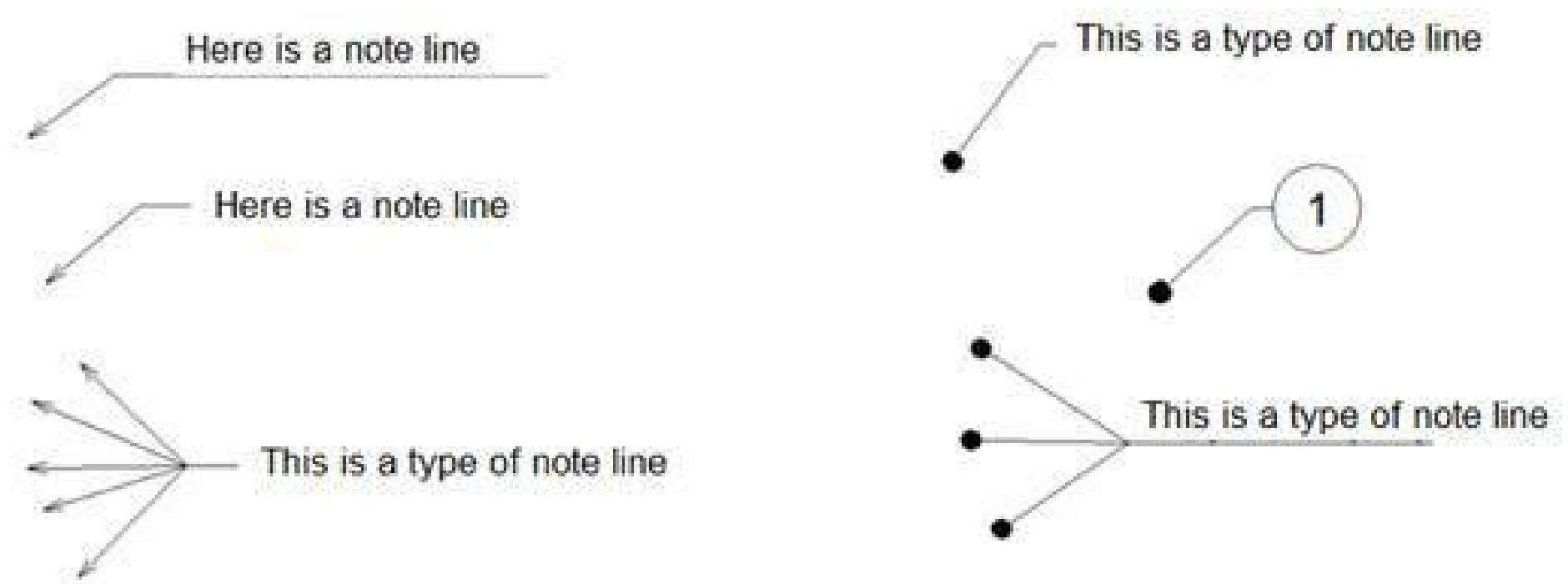
Chapter 15: Make A Note Line

Sometimes we need to create a note line to point to a certain point on the drawing to emphasize and attract special attention.

A note path includes the path and content of the note.

Of course, there can be multiple paths for the same note content.

Some common note line types.



The note line can be the name of the location marked with the excerpt, the position to be annotated, the position to be specially processed, the note about the similar objects, the numbering, or the highlight in the design. that readers understand need to grasp and obey.

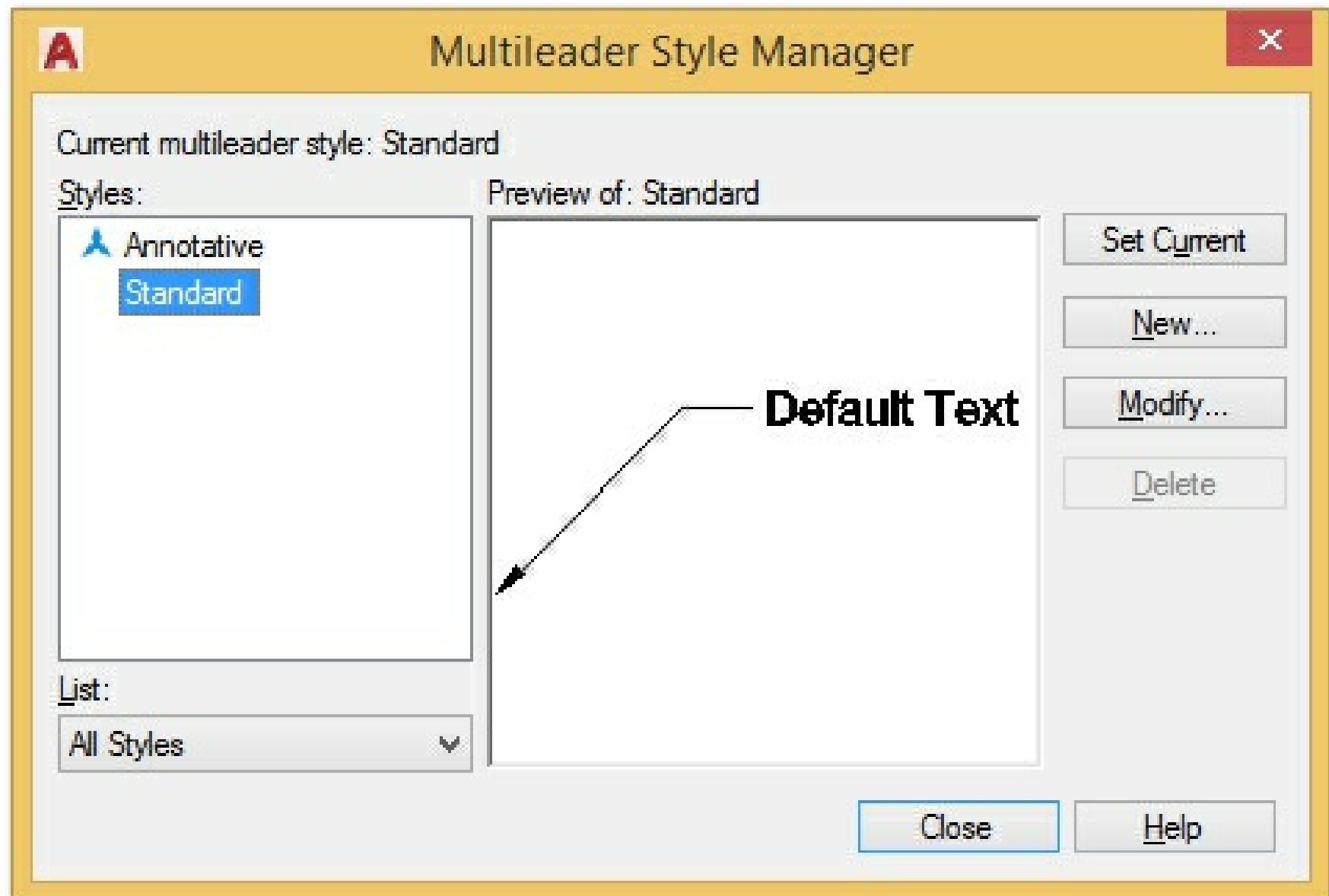
There are many ways to create path notes in AutoCAD. Create it manually by writing text and then using the Line command to drag it in. Or you can use one of the following 3 tools: Leader, Qleader, or Mleader.

Many people will like to use QLEADER (LE Command). But I recommend you to use MLEADER to create note lines.

To manage MLEADER, we will also create MleaderStyles to manage, similar to how we manage TextStyle or Dimstyle. And the MleaderStyles should also attach Annotative to insert them according to the drawing scale so that the appearance of the annotation line is guaranteed to be consistent between the scales after printing on paper.

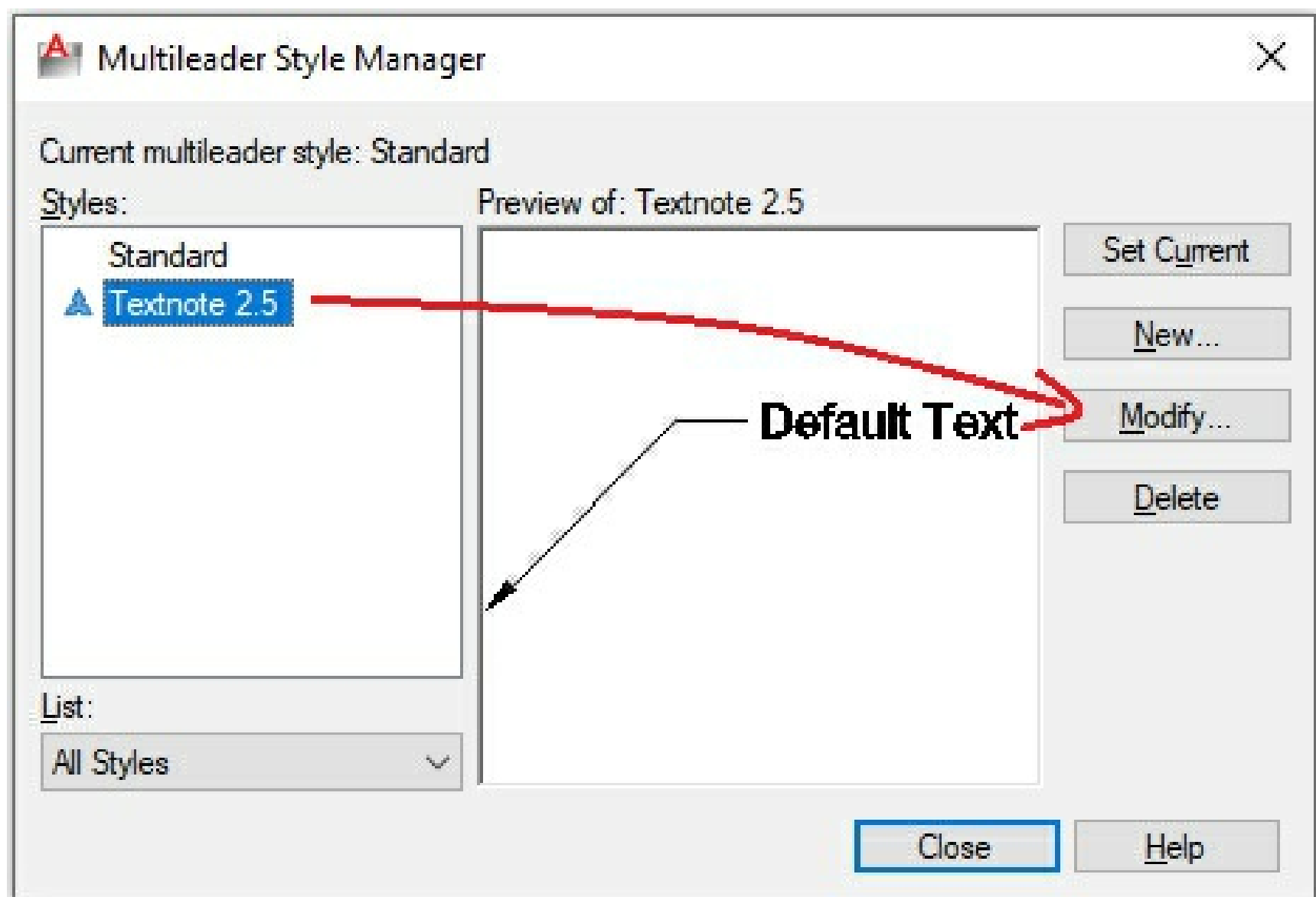
1. Create and manage MleaderStyle

- Type the command: **MLS**✓ to display the MleaderStyle management panel.

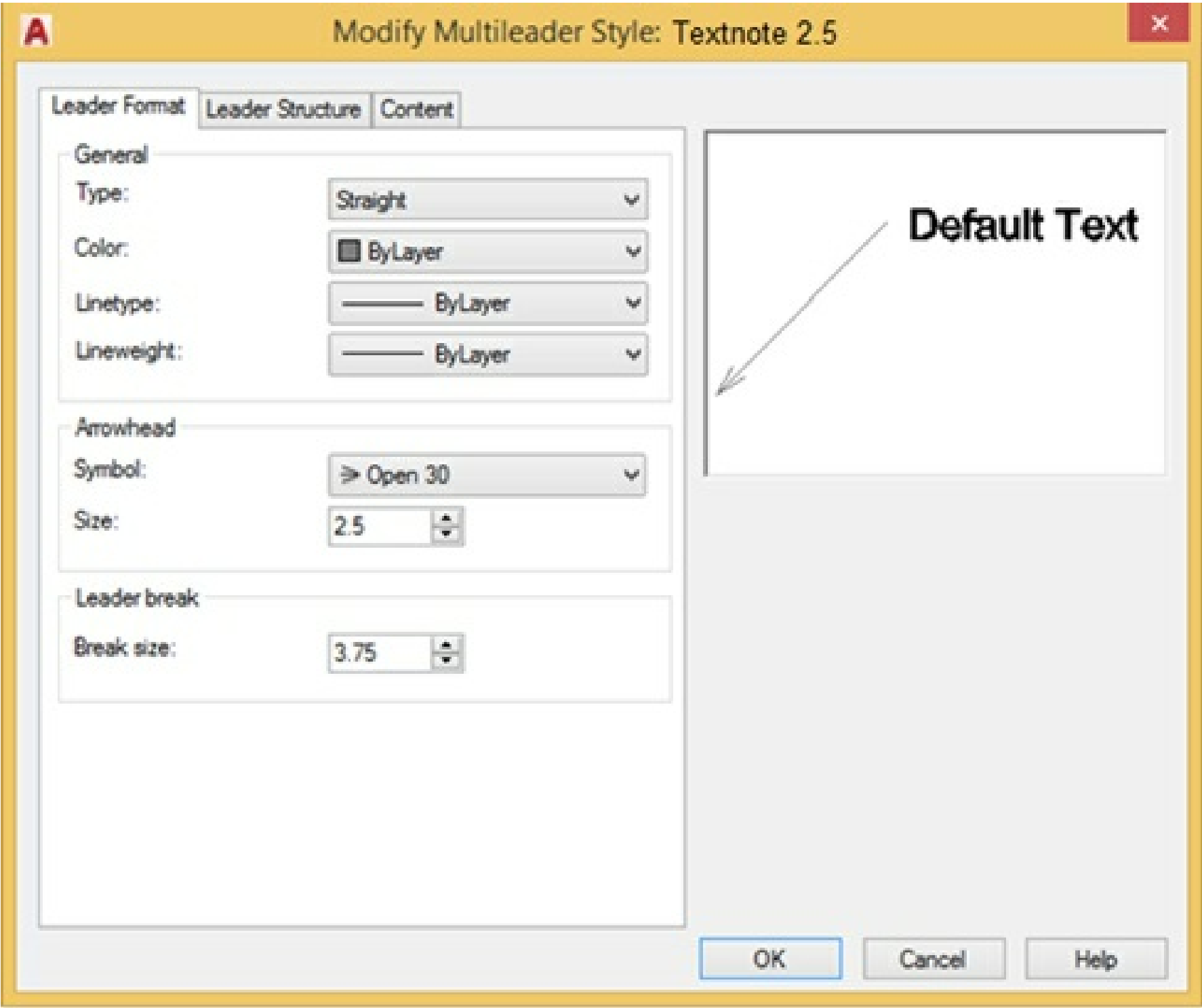


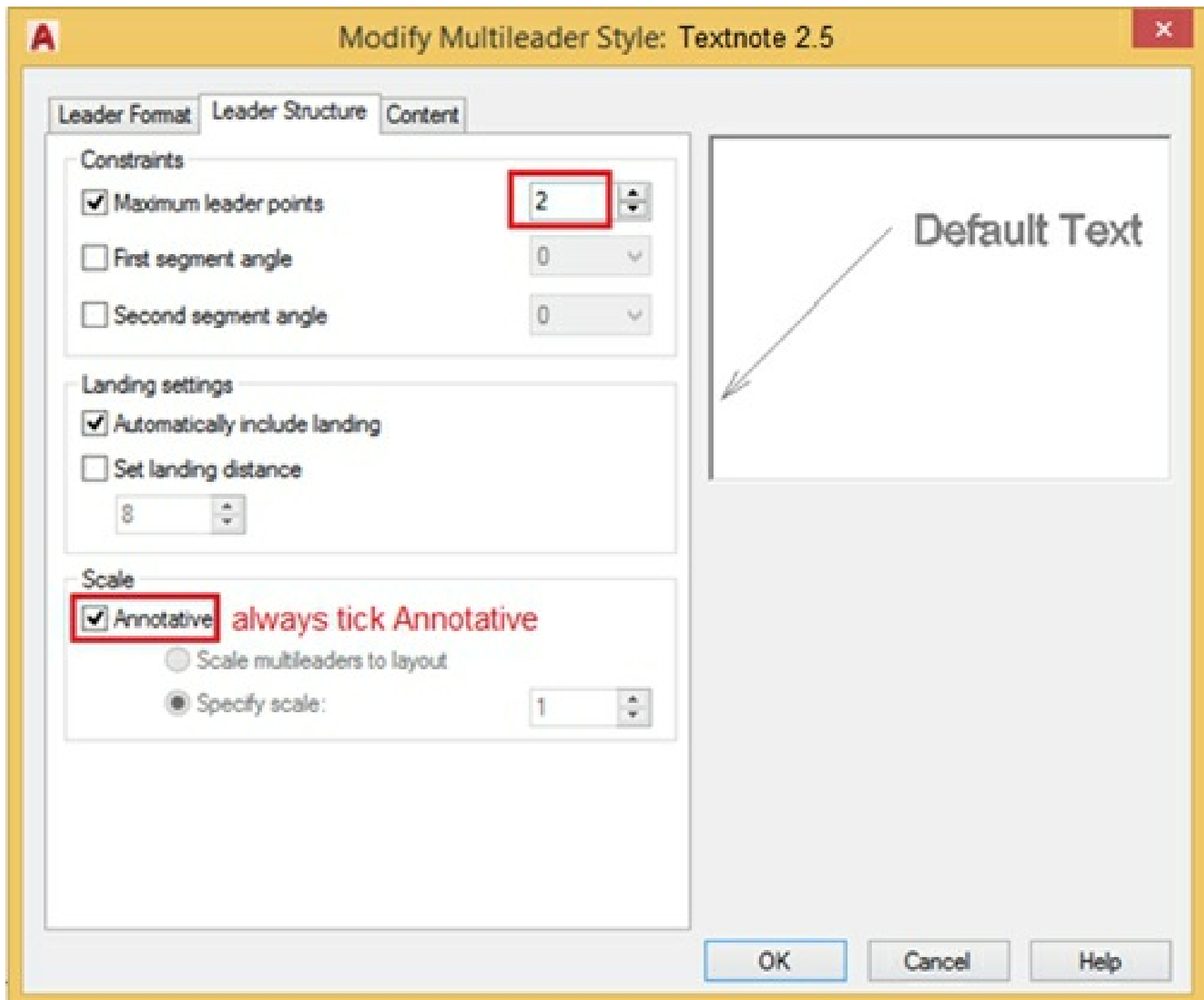
The external interface is quite similar to the Dimstyle management panel. And the fact that the management (create new, edit, delete, set style active) is the same as the Dimstyle manager.

- We won't use the Standard style, so leave it at that. Because you can't delete it.
- You will right-click on the Annotative style / Select Rename and change the name to "**Textnote 2.5**" with A4 size or "**Textnote 3.5**" for A3 paper. Then click Modify to edit the internal settings.

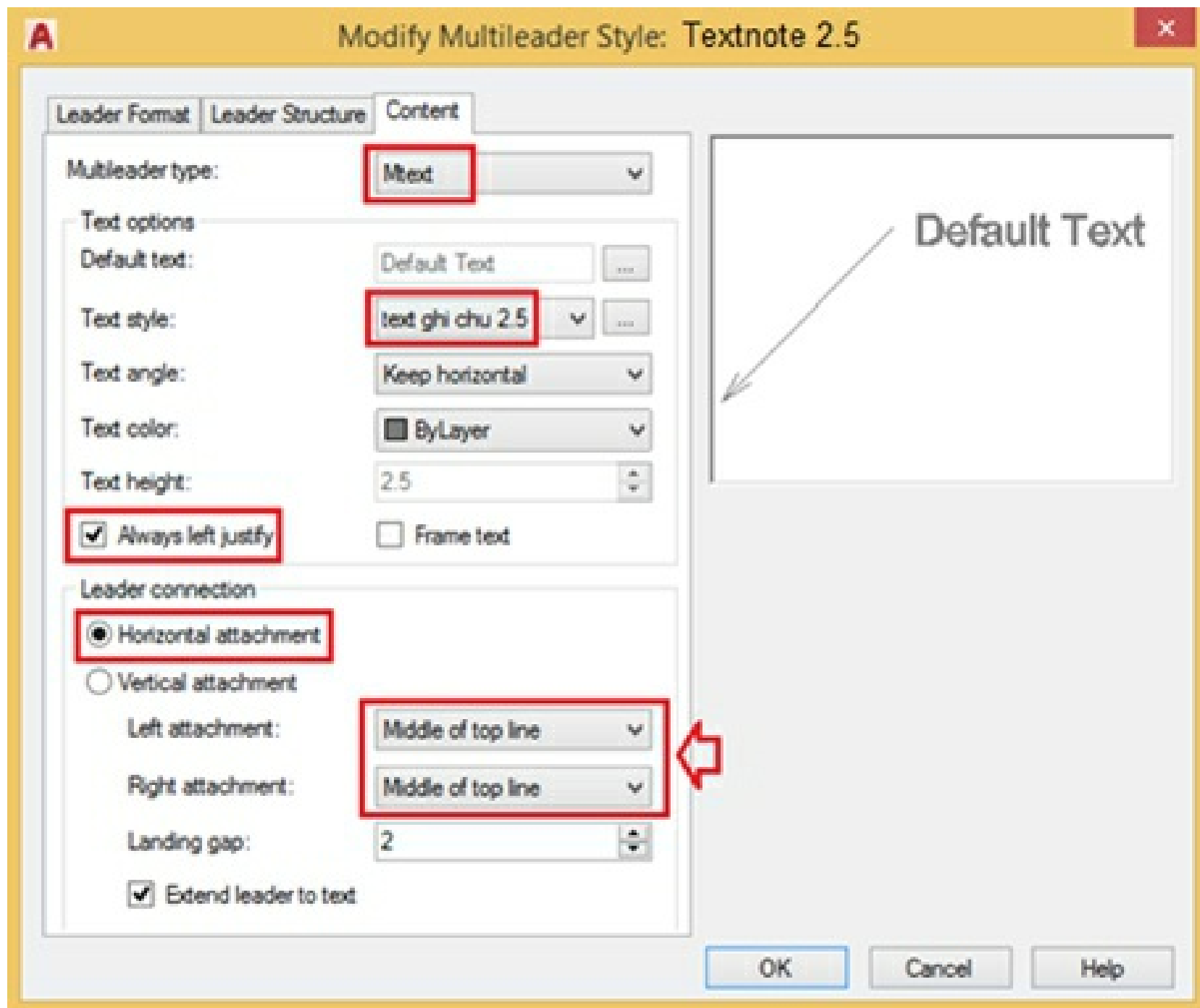


You can set it up like me:



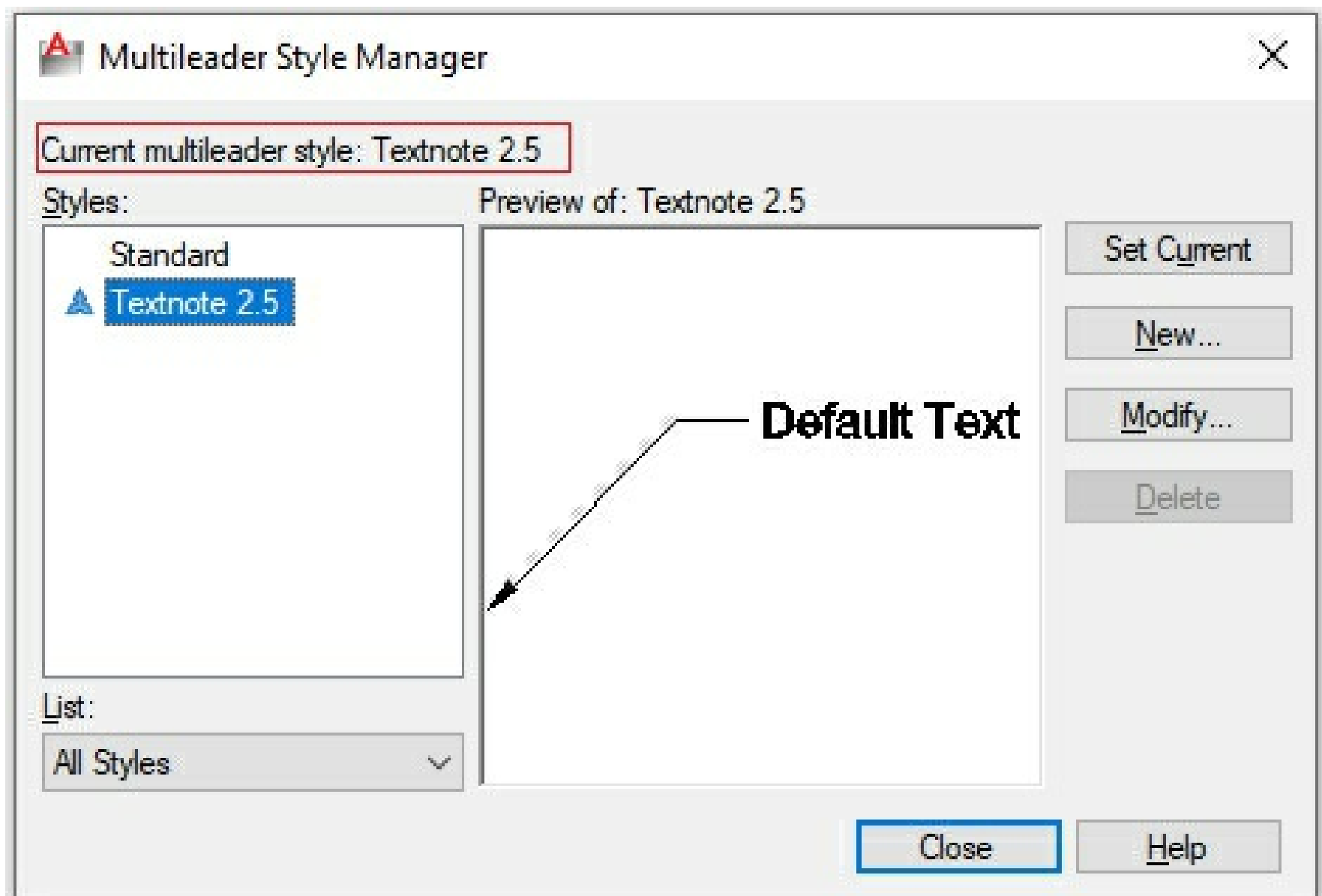


Note: the Text style selection for MleaderStyle, we also choose the "Textnote 2.5" style with A4 paper size or "Textnote 3.5" with A3 paper size. Here I am setting for A4 landscape size.



Then click OK to finish the installation.

Remember to double click on the “Textnote 2.5” style to set it as the current Mleaderstyle.

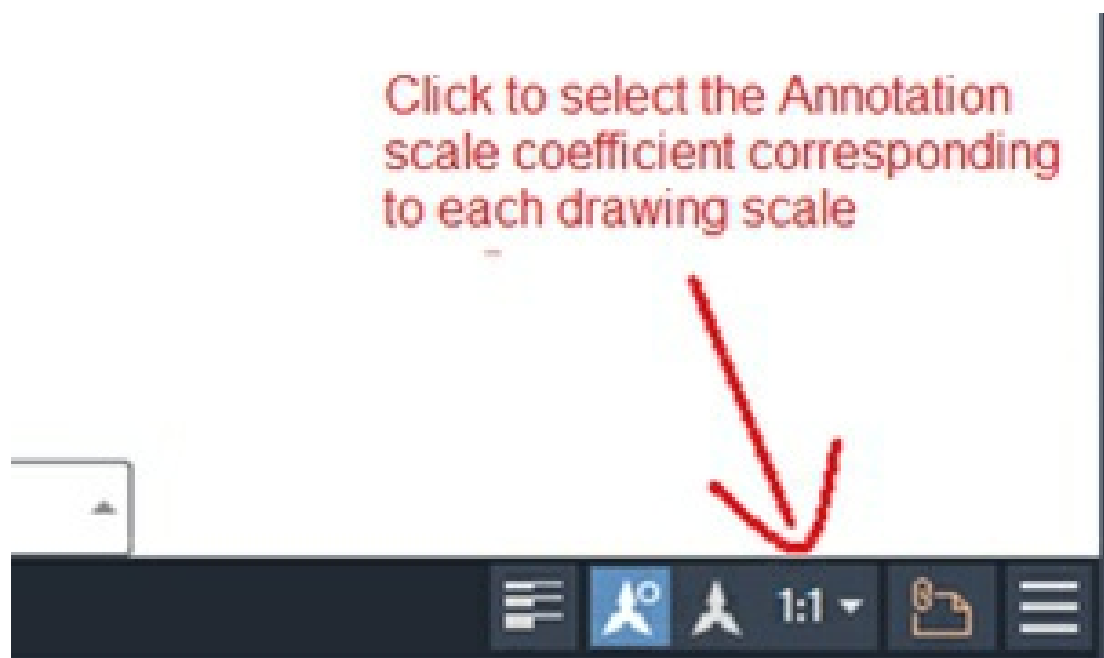


Then, on the screen, you can find a list of MleaderStyles on the Styles toolbar. Which style is visible means that the style is active, applicable to both TextStyle and DimStyle?



2. How to create note lines with MLEADER

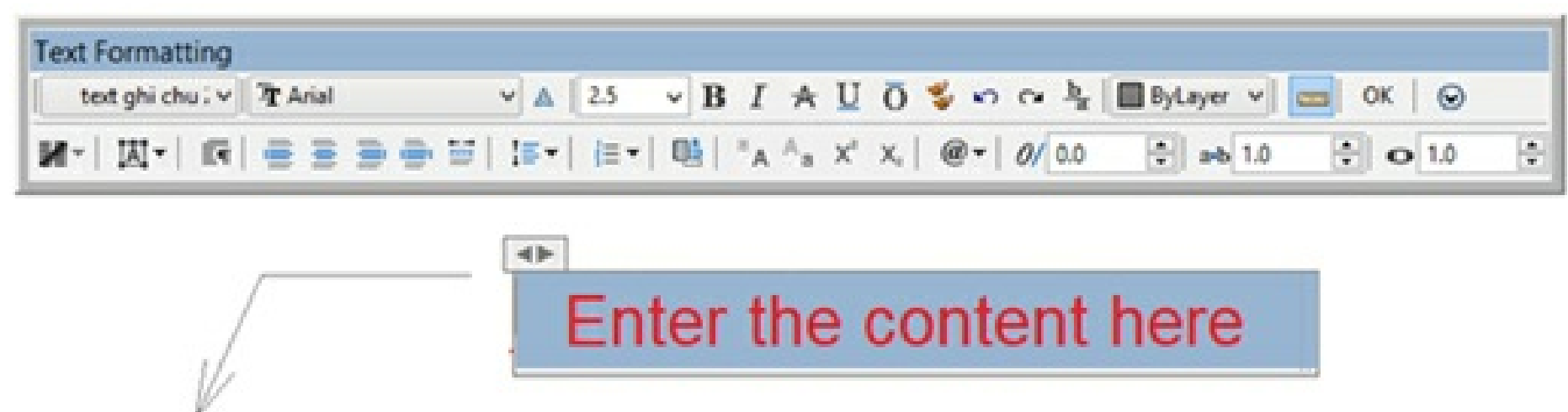
- Before creating a note line, you need to select the Annotation Scale coefficient corresponding to the drawing scale of each drawing that you are planning to create the note line.



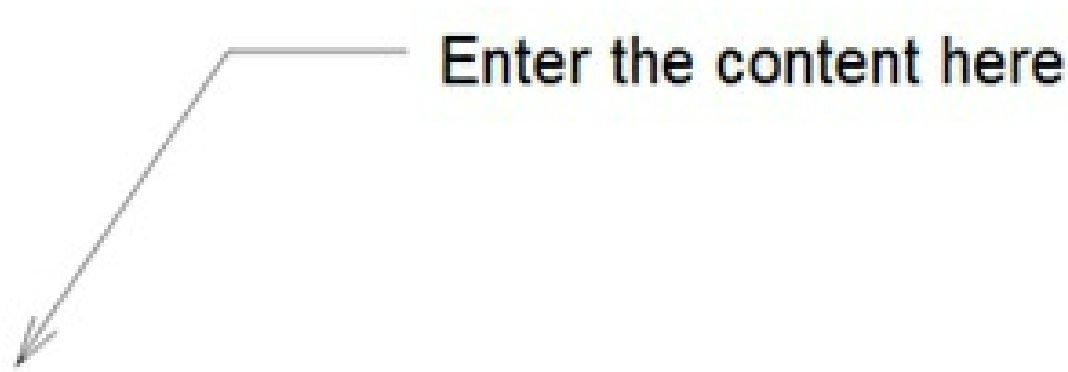
- ü If DRAWING SCALE = 1:1, you chose Annotation Scale = 1:1
- ü If DRAWING SCALE = 1:5, you chose Annotation Scale = 1:5

To create a note line, you type the command MLD↵.

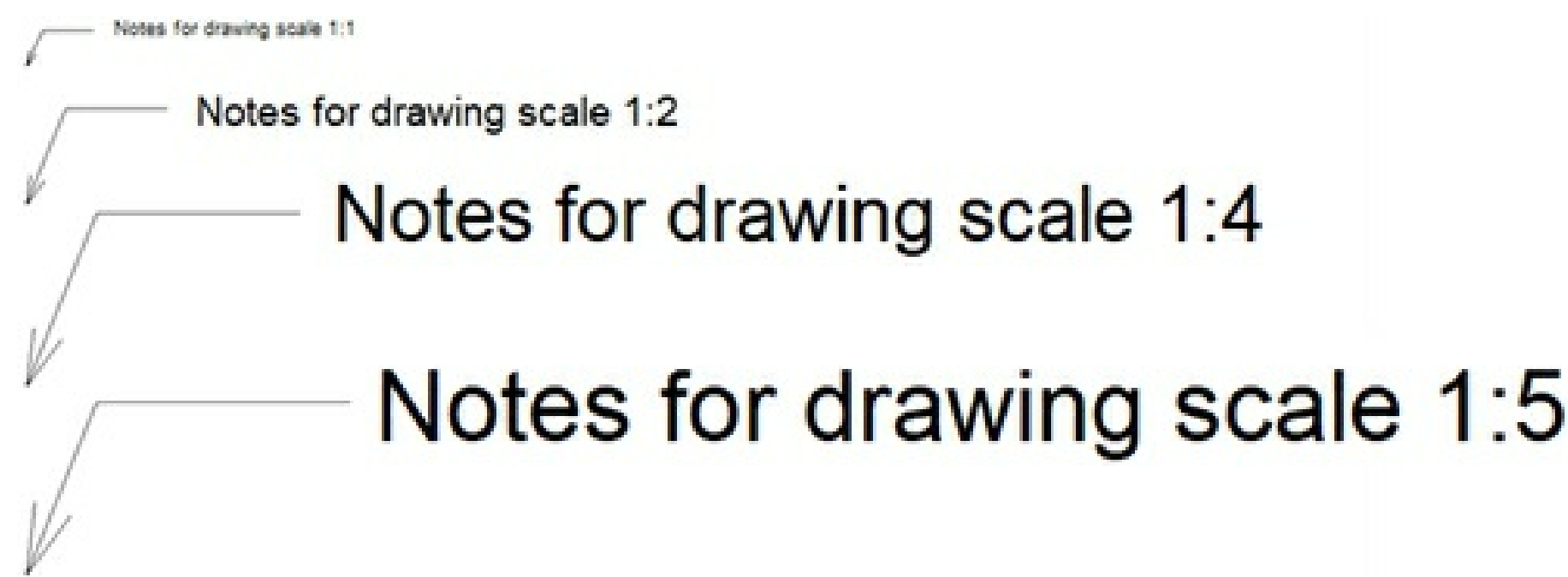
- Click the first point: the set point of the note arrow points to the position of the object to be noted
- Click the 2nd point: the set point of the starting position of the horizontal path of the note line
- Press Enter (can skip entering the size of the horizontal path _ landing line)
- Enter the content of the note in the Text Formatting bar.



To result as under here.

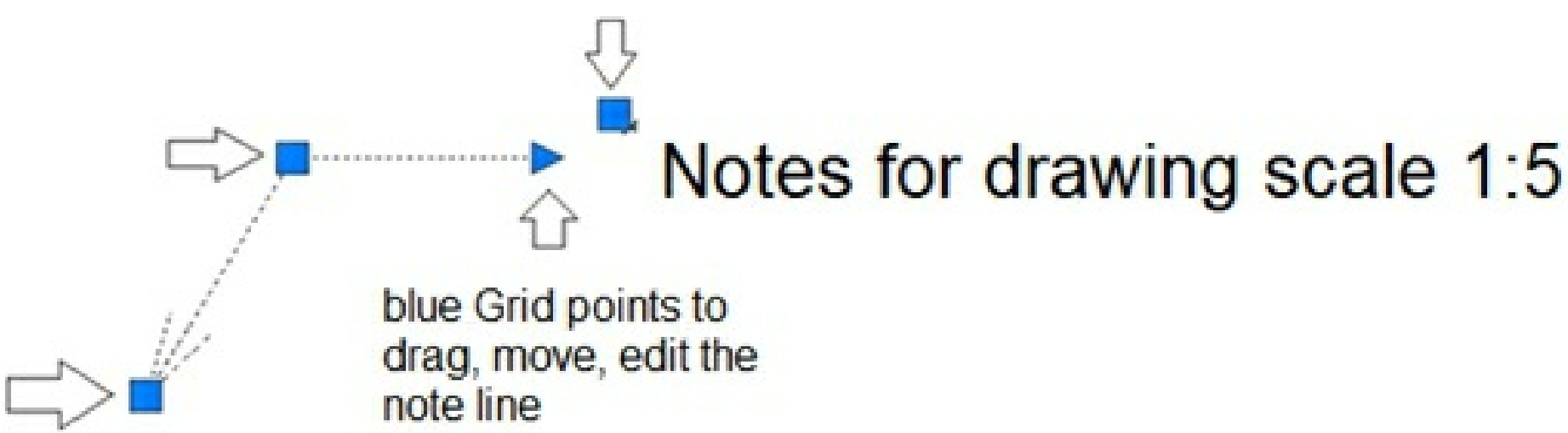


With the Annotative attached, you will see the difference between different DRAWING SCALE, the lines of notes have the same proportions.



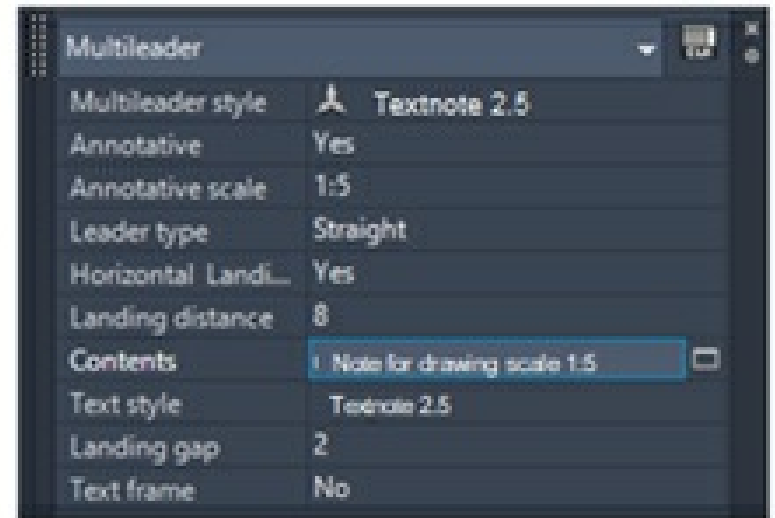
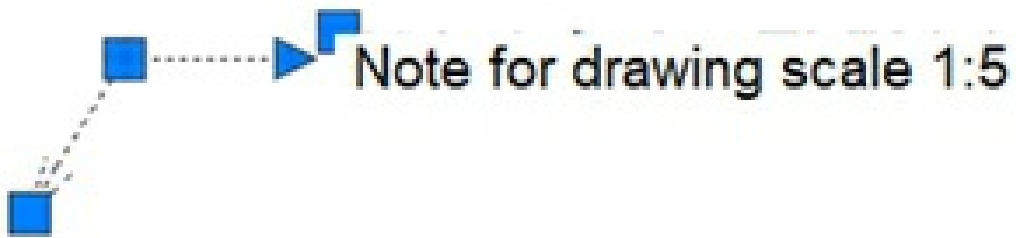
3. How to quickly edit a note line

When you click on any note line, you will see blue grip points appear. You can click on those grip points and move them to the desired position.



For deeper customization, you can customize it in the Quick Properties dialog box.

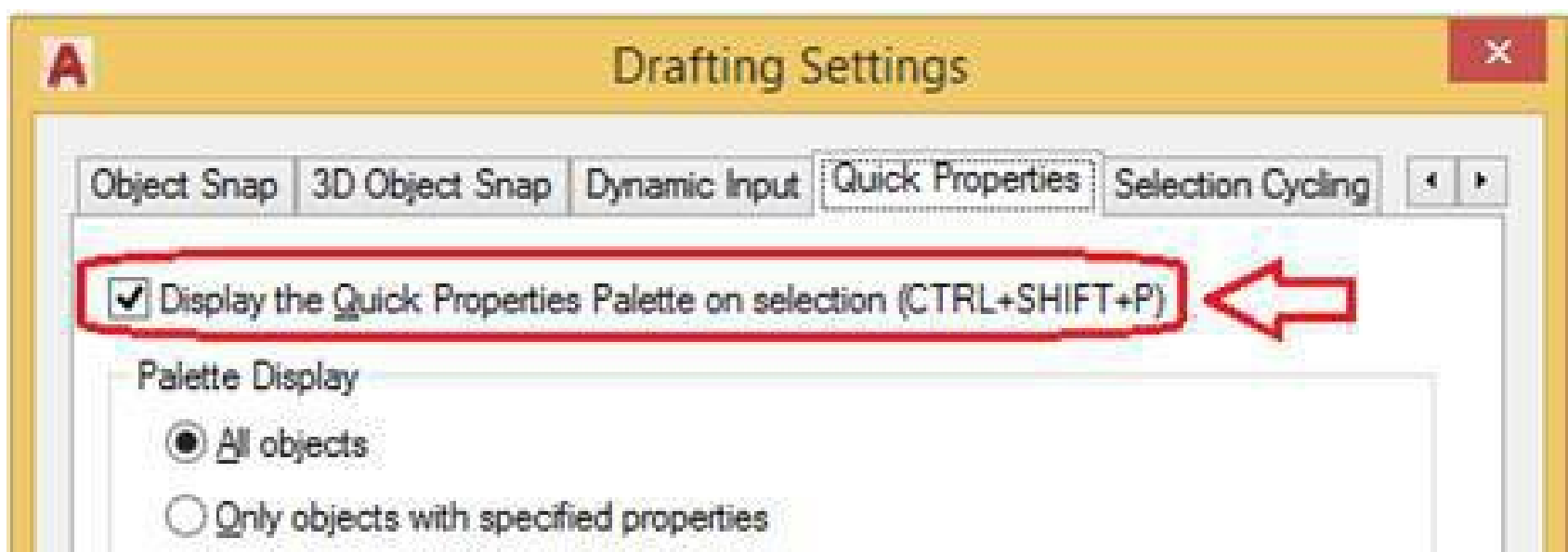
When you select any note line, a Quick Properties dialog box will appear.



In this dialog box, you can adjust a lot of parameters:

- Annotative scale: allows changing the Annotative scale coefficient of the line by adding a new coefficient and Deleting the old coefficient.
- Landing distance: Length of horizontal path
- Contents: allows to change the content of the note
- Landing gap: allows changing the distance from the end of the Landing line to the starting point of the note content

Well, did my friend know how to make the Quick Properties dialog box always appear when we select any object? Press “**Shift Ctrl P**”. It is located here:



Note:

If you want to align to the relative position of the note content with the horizontal path (Landing line), you must change the settings in MleaderStyle, not via Quick Properties.

Chapter 16: How To Synchronize The Scale Between Dashed Lines?

Almost all mechanical drawings have more or less dashed lines. e.g. centerlines, symmetry axes, extractor limits. Many with drawings showing hidden strokes. I see that not many people manage this category. As a result, on the same drawing after printing, there are sparse dashes, thick dashes, which are very unsightly, not to mention between different drawings.

If you go deep, there are a lot of things that are tied to the display of the dashed line (especially more complicated if printed in Layout). However, in this book, I will try to simplify it for you.

I only use dashed lines to represent centerlines, symmetry axes, line indicator lines, and contour lines. I never use dashed lines to represent hidden lines on an object's projection. The reason is that they just make the drawing more confusing. I try to project all the faces to see all the hidden edges. In case the hidden edge is deep inside, I will use a section or a separate section to represent it.

You can refer to my above method, the number of projections may be more, but the drawing is clearer, the reader will also be easier to visualize. Is that my own opinion? And how are you doing?

Can you write your method down here?

.....
.....
.....

Some people like to use the **LTSCALE** ✓ command to edit the dashed lines on the drawing. But this way is completely wrong because it does not completely solve the root problem. It's like you have a high fever but just pour water without taking antipyretic.

LTS affects the entire drawing, you can correct this drawing frame but will damage the other drawing frame, doing the front will fall behind. You cannot manage them. So the idea of using LTSCALE is a bad solution!

In addition, whether you choose to print on Model or Layout, the way to control the dashed line will also be different. Agree that, because we only print on the Model side, we also have our method. This method is a bit manual, but it will be effective.

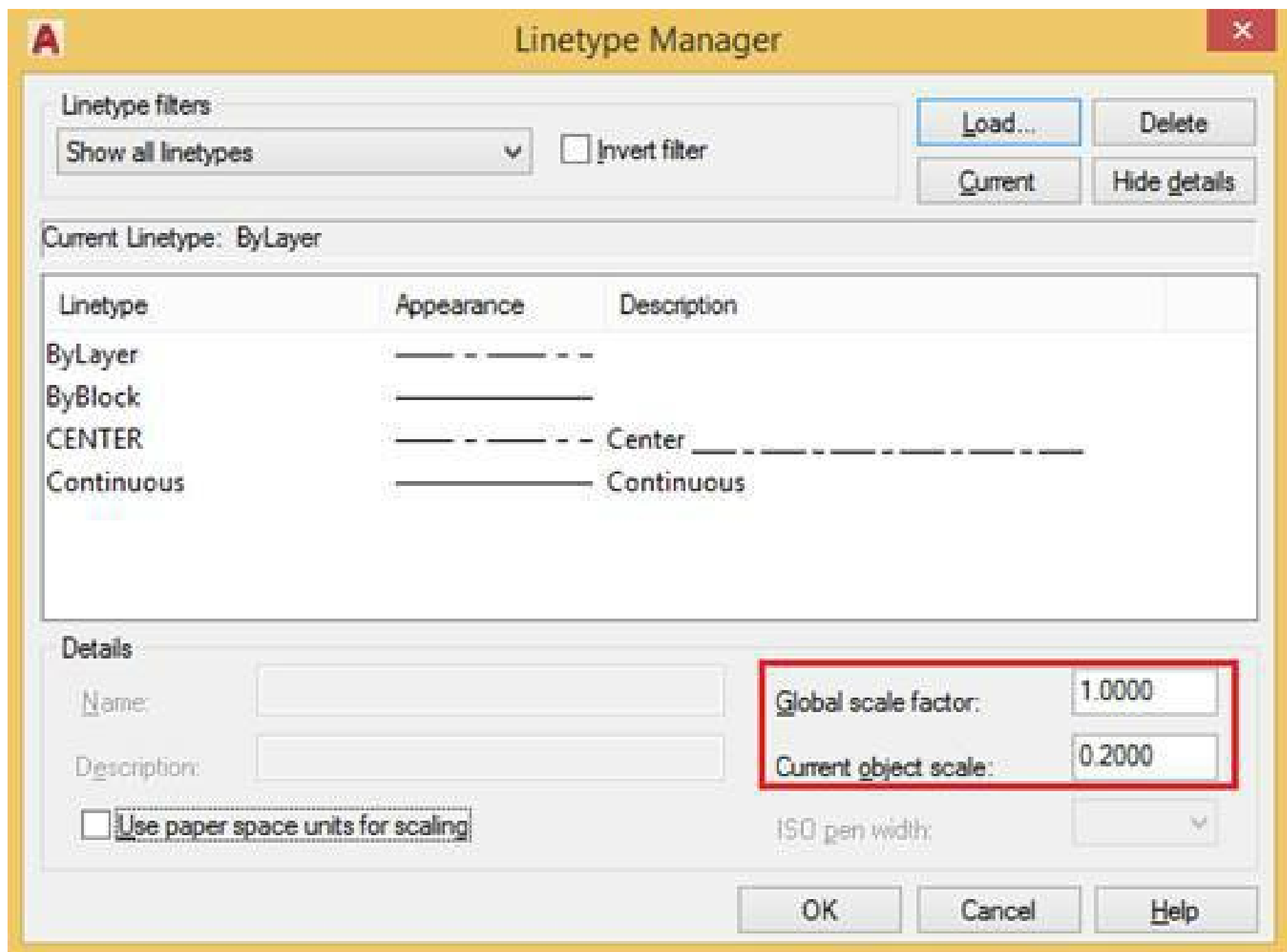
Please follow me!

- Type command **MSLTSCALE** ✓ , enter a value = 0 and press Enter.

(because if it is equal to 1 when you change the Annotation Scale, the scale of the dashed lines will automatically change, losing control in the Model)

- Type the command **LT** ✓ and set 2 values as follows:

- **Global-scale factor** (it is essentially the LTSCALE variable): set the value = 1
- **Current Object Scale** (it is essentially a CELTSCALE variable): set a value of your choice, it is a discontinuous scale with DRAWING SCALE 1:1, as long as you see the break as pleasing to the eye, here with the size A4 I choose the value is 0.2
- **Use paper space units for scaling:** this item only makes sense if printed on Layout, so here we consider it meaningless because it only prints on Model.



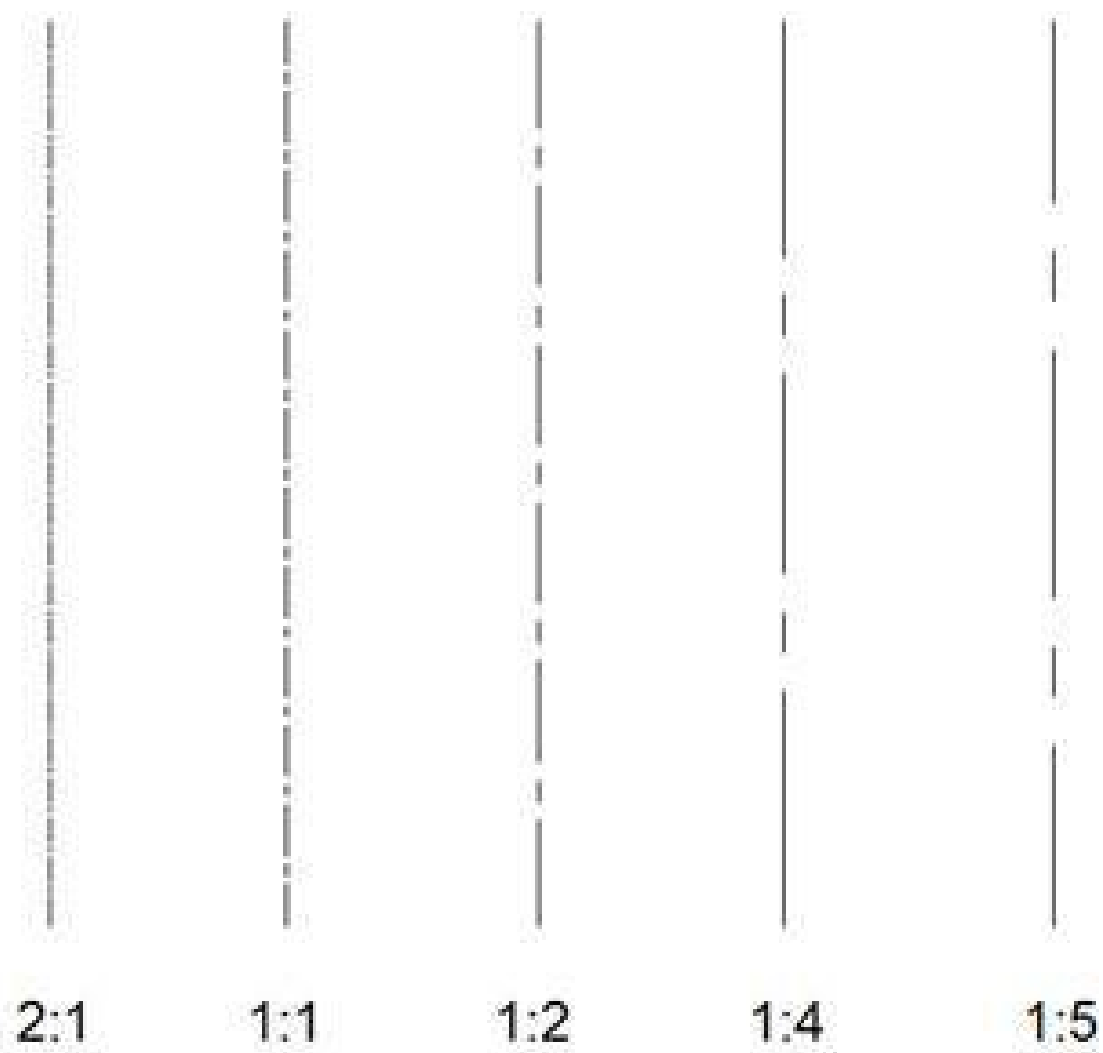
1. Principle of showing dashed lines on multi-scale drawings

Principle: ***"On the same paper, all drawings, no matter what scale is printed, the ratio between the dashed lines must be the same"***.

Let's say I take the "stretch factor that determines the break on A4 paper size 1:1 DRAWING SCALE" as the base value and choose that value to be 0.2

Then, for drawings with other DRAWING SCALE, the scaling factor will scale by a multiplier of 0.2

- DRAWING SCALE 1:2 => expansion factor = $0.2 \times 2 = 0.4$
- DRAWING SCALE 1:4 => expansion factor = $0.2 \times 4 = 0.8$
- DRAWING SCALE 1:5 => expansion factor = $0.2 \times 5 = 1$
- DRAWING SCALE 2:1 => expansion coefficient = $0.2 / 2 = 0.1$



Dotted line stretch according to drawing scale

Note: Thickness of dashed line = thickness of the basic stroke

We specify this density in the Layer Settings panel.

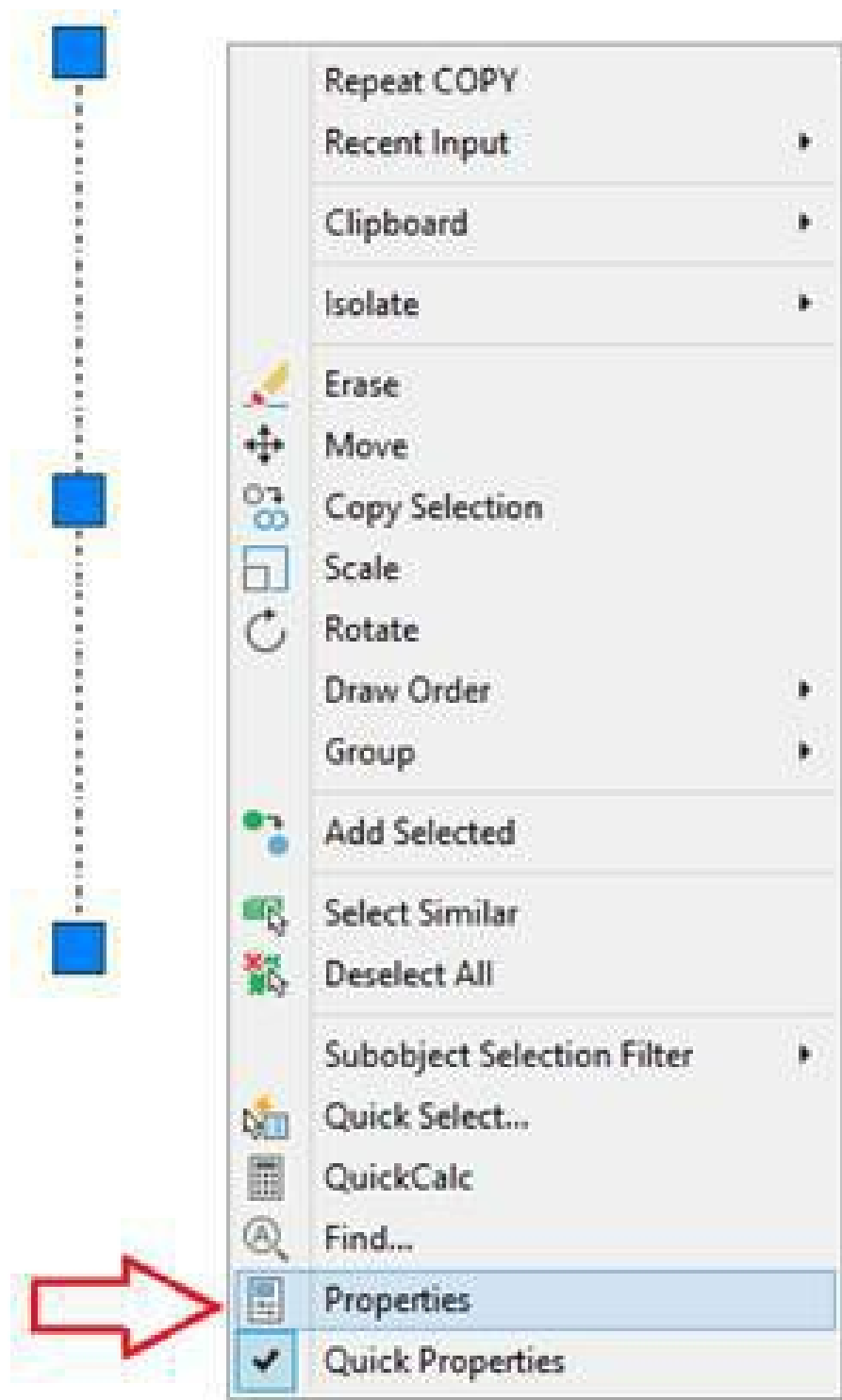


2. Handling dashed lines on files with multiple drawing scales

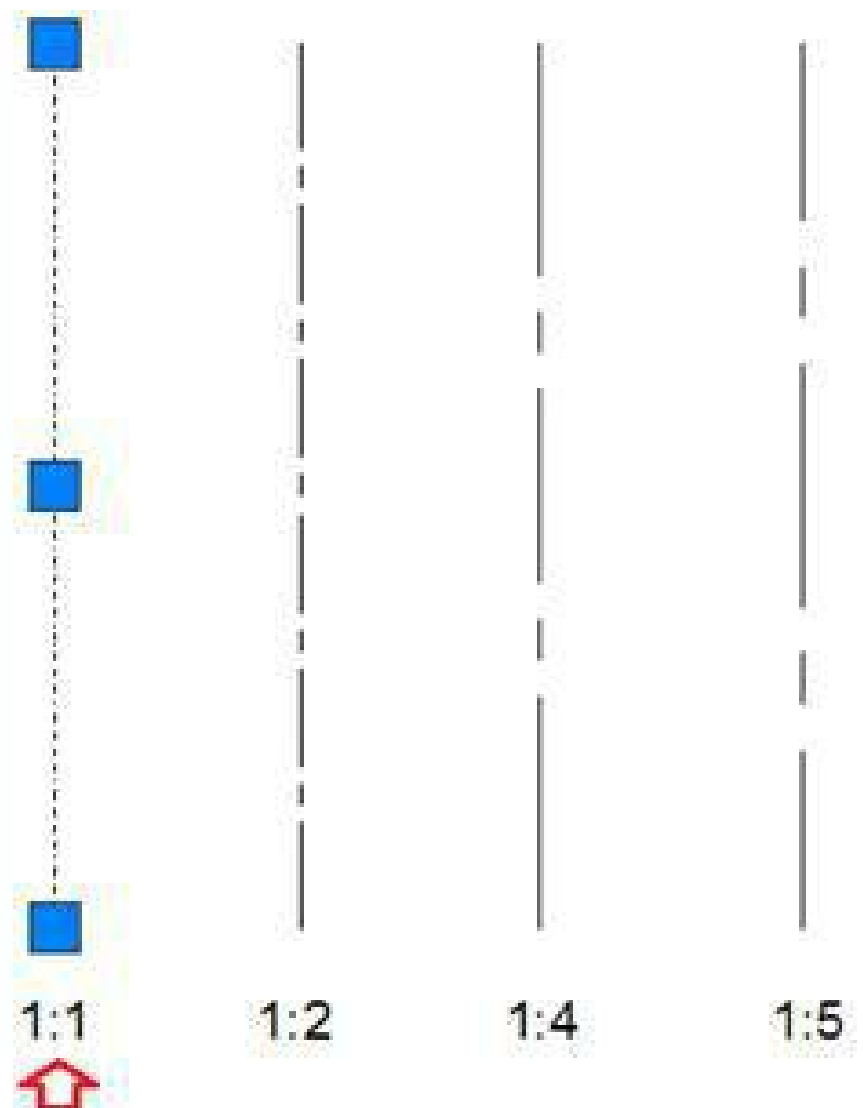
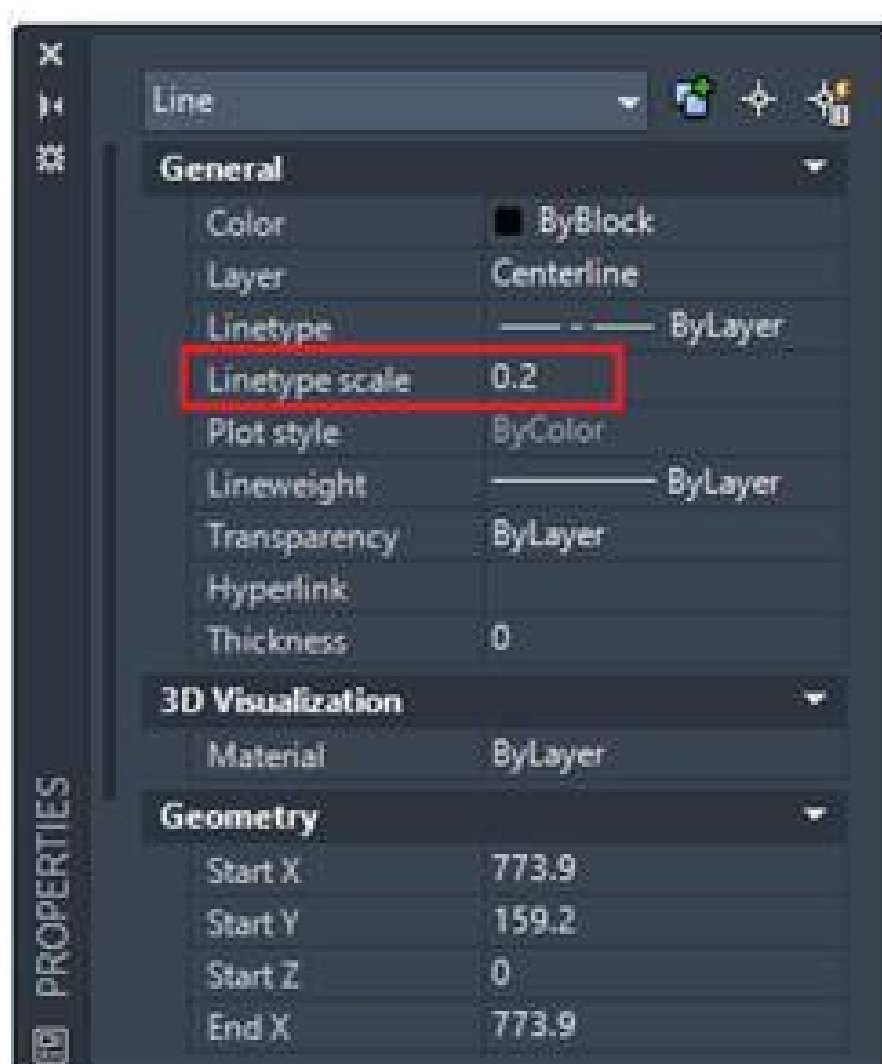
By taking the value 0.2 of the 1:1 drawing scale as the basis, with other drawing scales we just multiply by a factor of 0.2 and set the Linetype scale property to that dashed line value, corresponding to each drawing scale

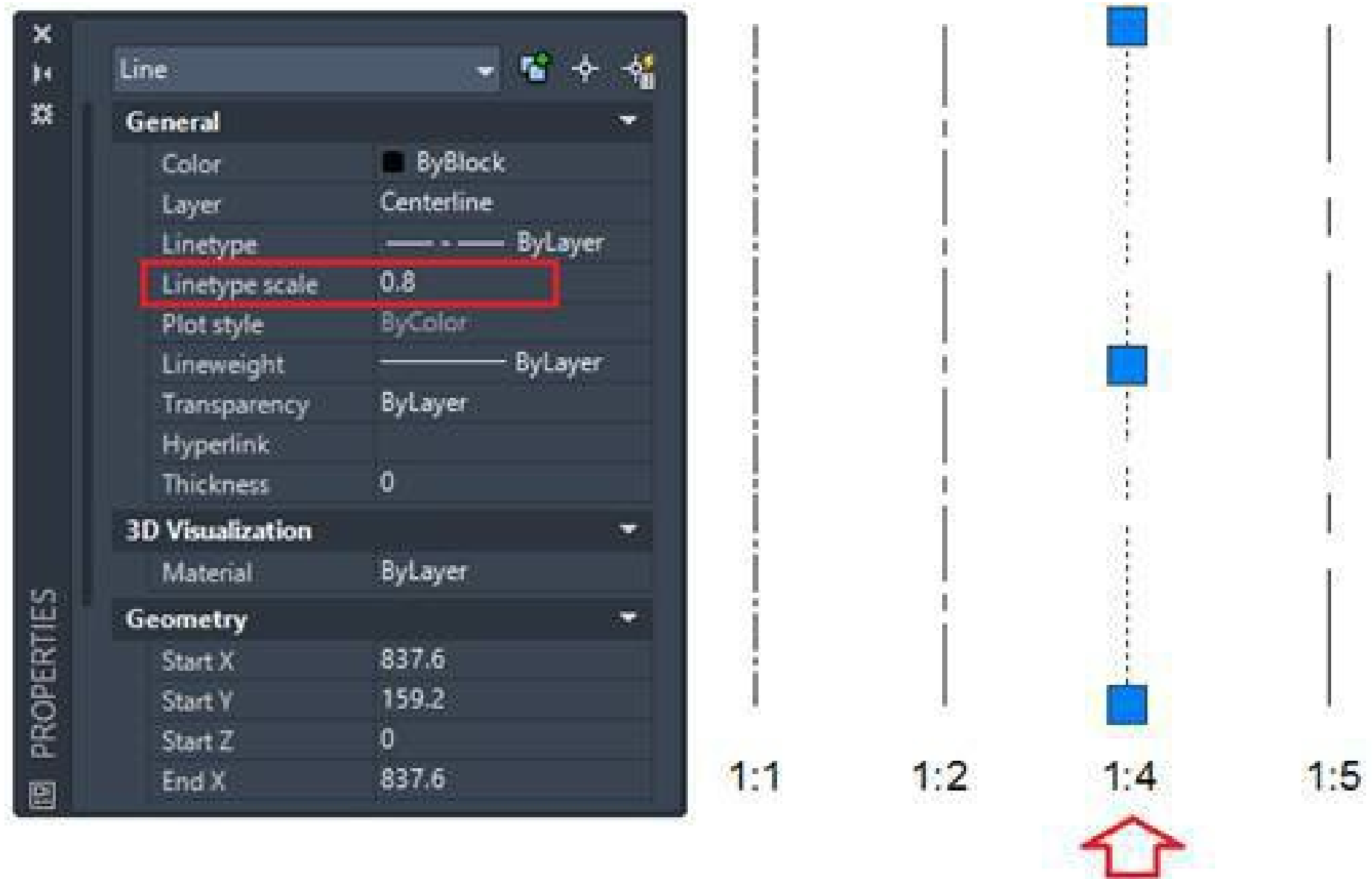
With any dashed line, to set the Linetype scale value for it, we do the following:

- Select object
- Right-click, select Properties (or type PR ↵)



- Enter the corresponding number (according to the drawing scale) in the *Linetype scale*





This way, even though the drawings are different in scale, when you print them out, the dashed lines will always be in sync with each other in scale. Beautiful!

Note: You can also use the same method for Hatch strokes. However, for small sections, you may not need to follow the rules.

Chapter 17: Mass Printing Drawings In Autocad

Have you ever applied for approval of drawings that need to be printed in tens or even hundreds?

Do you have to do the same thing over and over again: call the print job and set the print area for each of those drawings, dozens, hundreds of times over hours?

Impatient and boring, right?

And it takes too long if you have a long line of other things to do.

So is there a faster way?

Yes!

This weapon is extremely powerful, it is like it has been programmed and your job is just to press the trigger button. Very easy to operate and effortless.

Imagine just one print job that you can print all hundreds of drawings with unlimited quantity. You can take advantage of going to get a drink or sip a few sips of coffee while you wait to get a whole bunch of printed drawings in sync and to the right scale.

Of course, very few people know how to do this, because it is created by professionals and only for professionals. But it's about to stop being a secret to you.

But before I use it, I want you to understand some of the basics of printing a drawing in the most convenient way you're used to.

Wait and see, do you find anything new here?

1. Print a drawing in Model

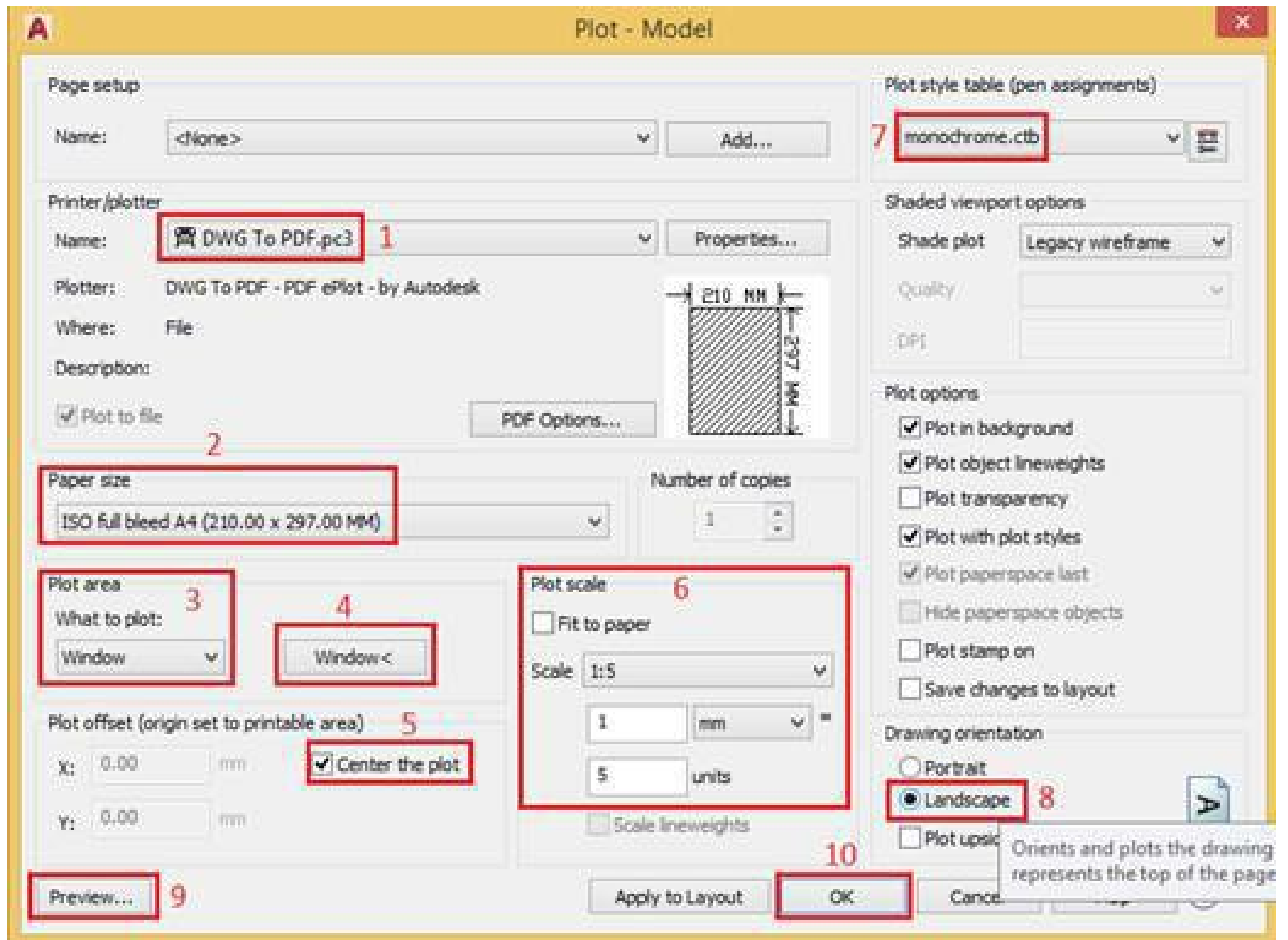
1. 1. To print any drawing in Model space, do the following:

- Press *Ctrl P*

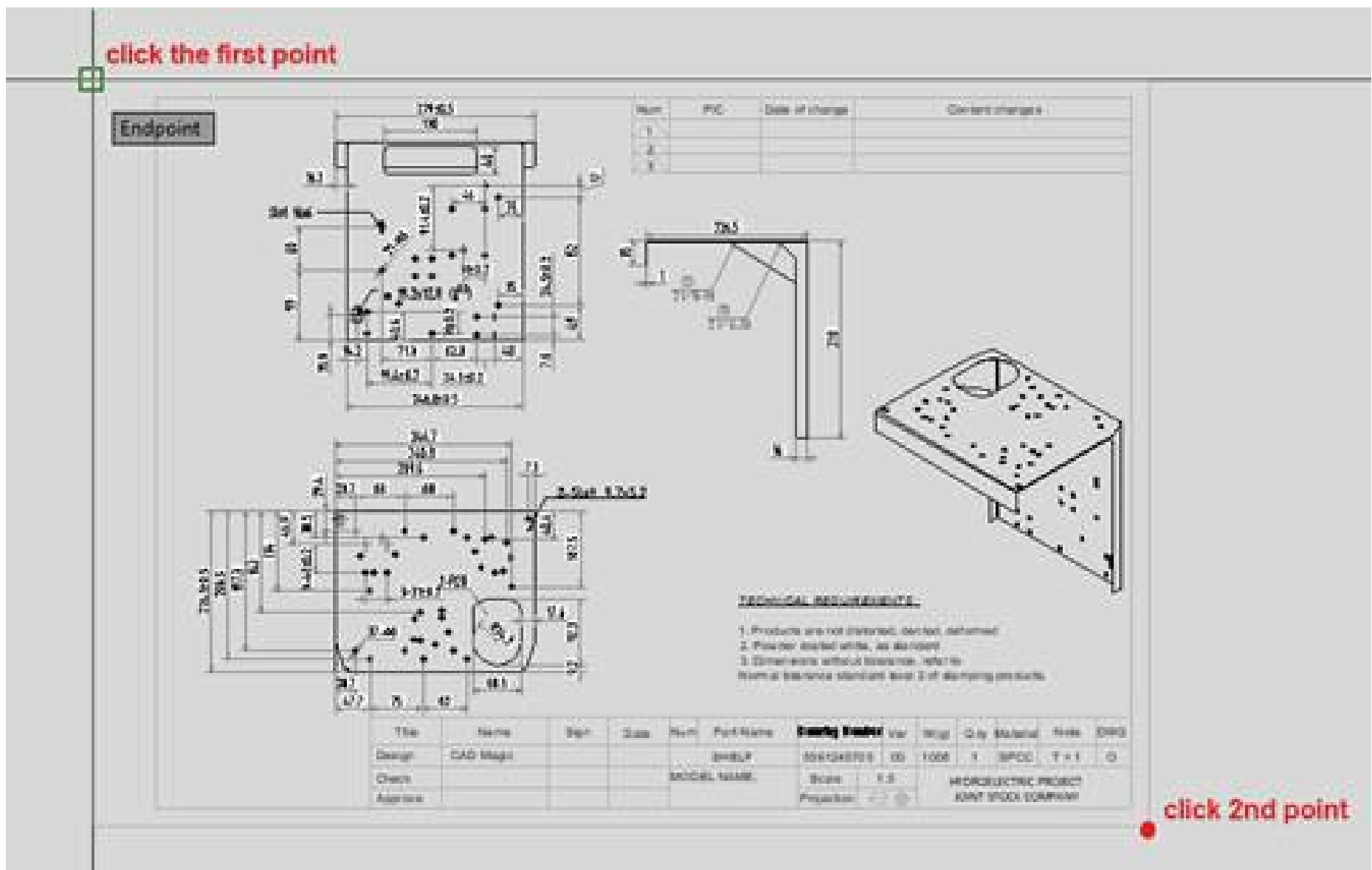
Step 1: Select the printer. Be it a real printer in your office or a PDF printer. Here I choose a PDF printer (DWG to PDF.pc3) because the computer I work on at home does not have a real printer.

Step 2: Choose the paper size

Step 3: Select the format type for the print area as “Window”



Step 4: Click on Window, scan the printed area as 2 diagonal points of the title block



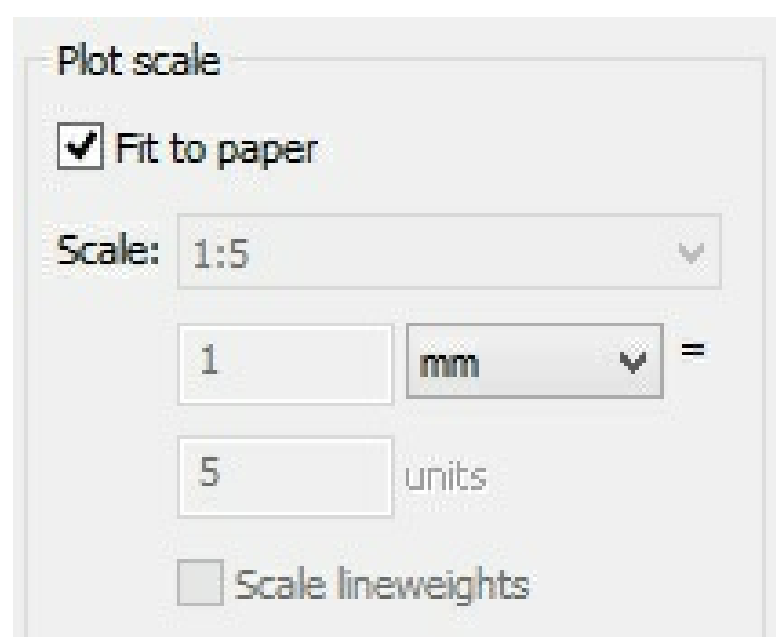
Step 5: Tick **Center to plot** to print in the center of the paper

Step 6: Select drawing scale

If the drawing you are going to print has a 1:5 drawing ratio, here you also choose a 1:5 scale. (Similar to other ratios)

Note: You can select Fit to paper but make sure that the other scale number does not change after you tick Fit to paper.

For example, if Custom is 1:5 (your drawing scale is exactly 1:5) but when you check Fit to paper it still shows 1:5, you can choose Fit to paper and print it out. Make sure the drawing scale is correct 1:5.

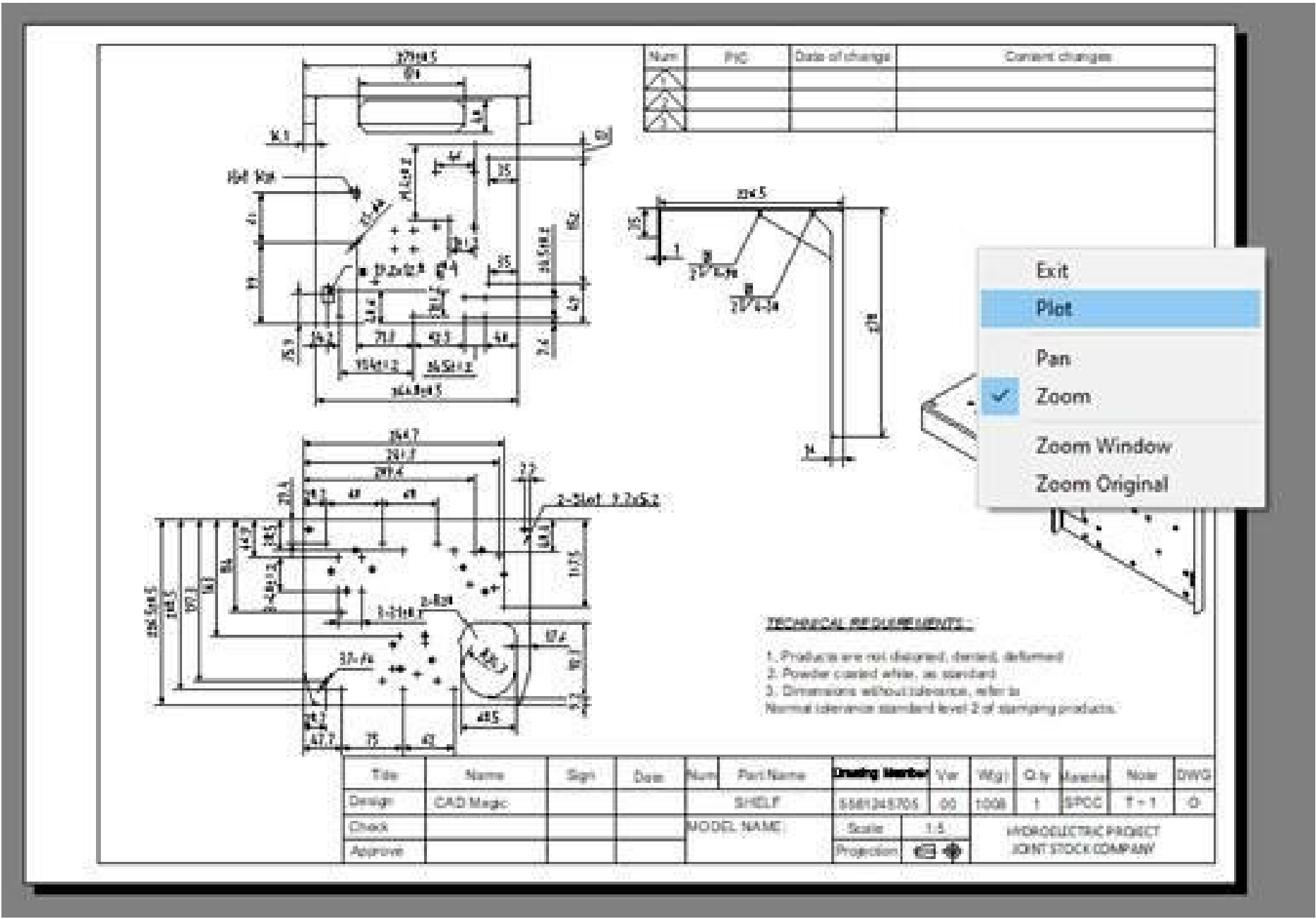


But usually, when you click on Fit to paper, the scale will be changed slightly (for example, if 1:5 it will become 1:5.125, and make the scale after printing will be slightly wrong. The cause is only the print frame setting, I will guide you to fix this in the next section.

Step 7: Select the print stroke, usually, we choose monochrome.ctb, that is, print in black and white (if you want to print in color, you choose acad.ctb). You don't care about resetting the print anymore, because we print in layers, the thickness, thickness, lightness of each line type is already specified in the layer management panel.

Step 8: Select the paper orientation. Usually, horizontal printing, choose Landscape. The portrait is vertical print

Step 9: Click Preview to see the drawing before printing



Step 10: After Preview first, if you see that the drawing is satisfactory, then right-click on the screen, select Plot to print the drawing to paper.

In the case of PDF printing, AutoCAD will ask what you want to name the file and where to save it.

And that's it!

1.2. How to print the correct scale of the drawing?

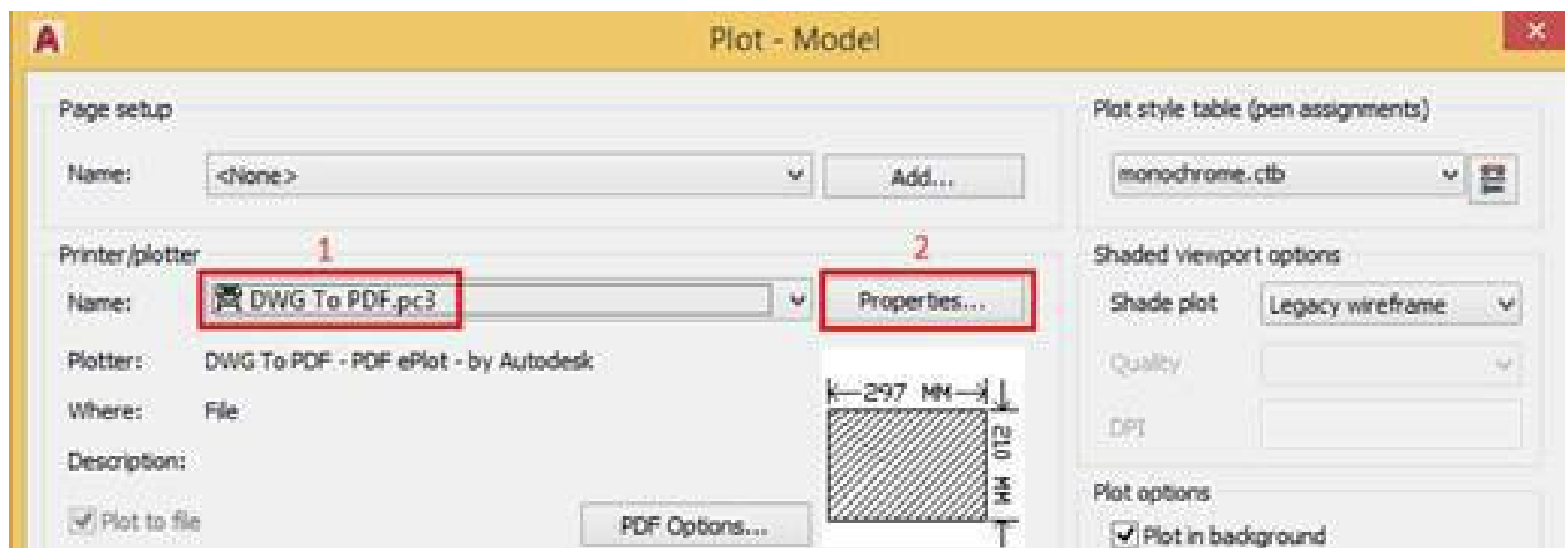
Today, the standard printing of the drawing scale is no longer so important because the reader of the drawing does not use a ruler to calculate it anymore, but can measure it directly on the 2D drawing. Even the ratio of drawings is wrongly recorded by many readers when they do not care. However, technically you shouldn't accept that. Holding a properly presented and printed drawing, you will feel more confident, definitely happier than holding a drawing that you just printed out for reference.

I will show you how to print every drawing at 100% scale (even with mass printing). Those who say that they can't print at the correct 100% rate just don't know-how. And whoever says that choosing Fit to paper will not print the exact scale is also a misunderstanding. I call this how to print borderless or print in the correct frame (the printing frame is the size of A3, A4, ...).

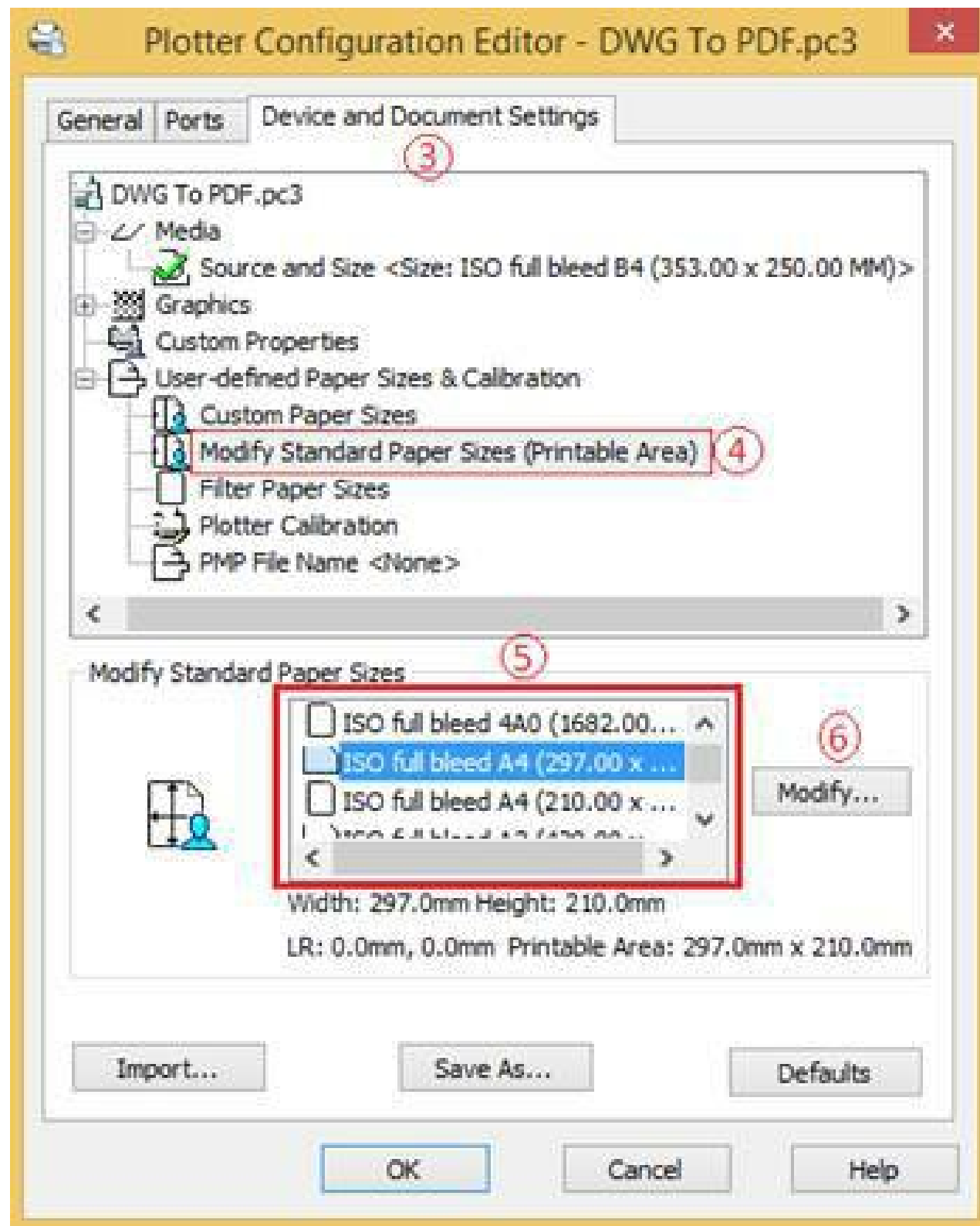
Put it like this: The default rule of the printer will always subtract 2-4mm paper margins, depending on the type of printer. For example, if you print A4 size of 297x210mm, by default (if you do not interfere), it will print on a paper area slightly smaller than 297x210mm, ie the drawing scale will deviate a bit.

To print standard scale, you need to print border overflow (overflow), bring all 4 paper margins (top, bottom, left, right) to zero. We do the following:

- Press Ctrl P to reopen the Plot dialog box.
- Select your actual printer name (1) (here I'm still leaving the PDF printer, it's okay) then click Properties (2)



- Click **Device and Document settings** (3)
- Select **Modify standard paper sizes** (4)
- In (5), drag the vertical slider to find the size of the paper you plan to print (A3 or A4, ...), click on that size, and then click Modify (6).



Note: The Custom properties section is the setting to configure the printer's driver, allowing you to choose single-sided, 2-sided printing, select paper trays, ... but this part you should leave as default, ie ignore.

- Set all numbers in (7) to zero. Then select Next (8)

Custom Paper Size - Printable Area

Begin

Media Bounds

▶ **Printable Area**

Paper Size Name

File name

Finish

The Preview tile indicates the printable area based on the currently selected paper size. To modify the non-printable area, adjust Top, Bottom, Left and Right edges of the page.

NOTE: Most drivers calculate printable area from a specific measurement away from the edge of the paper. Some drivers, such as Postscript drivers, measure printable area from the actual edge of the paper. Verify that your plotter is capable of plotting from the actual dimensions you specify.

Top :

Bottom :

Left :

Right :

0

0

0

0

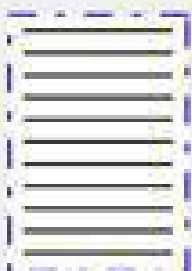
↑

↓

↑

↓

Preview



8

< Back
Next >
Cancel

- Name the printer with your standard print style just created. Then click Next (10)

Custom Paper Size - File name

Begin

Media Bounds

Printable Area

Paper Size Name

▶ **File name**

Finish

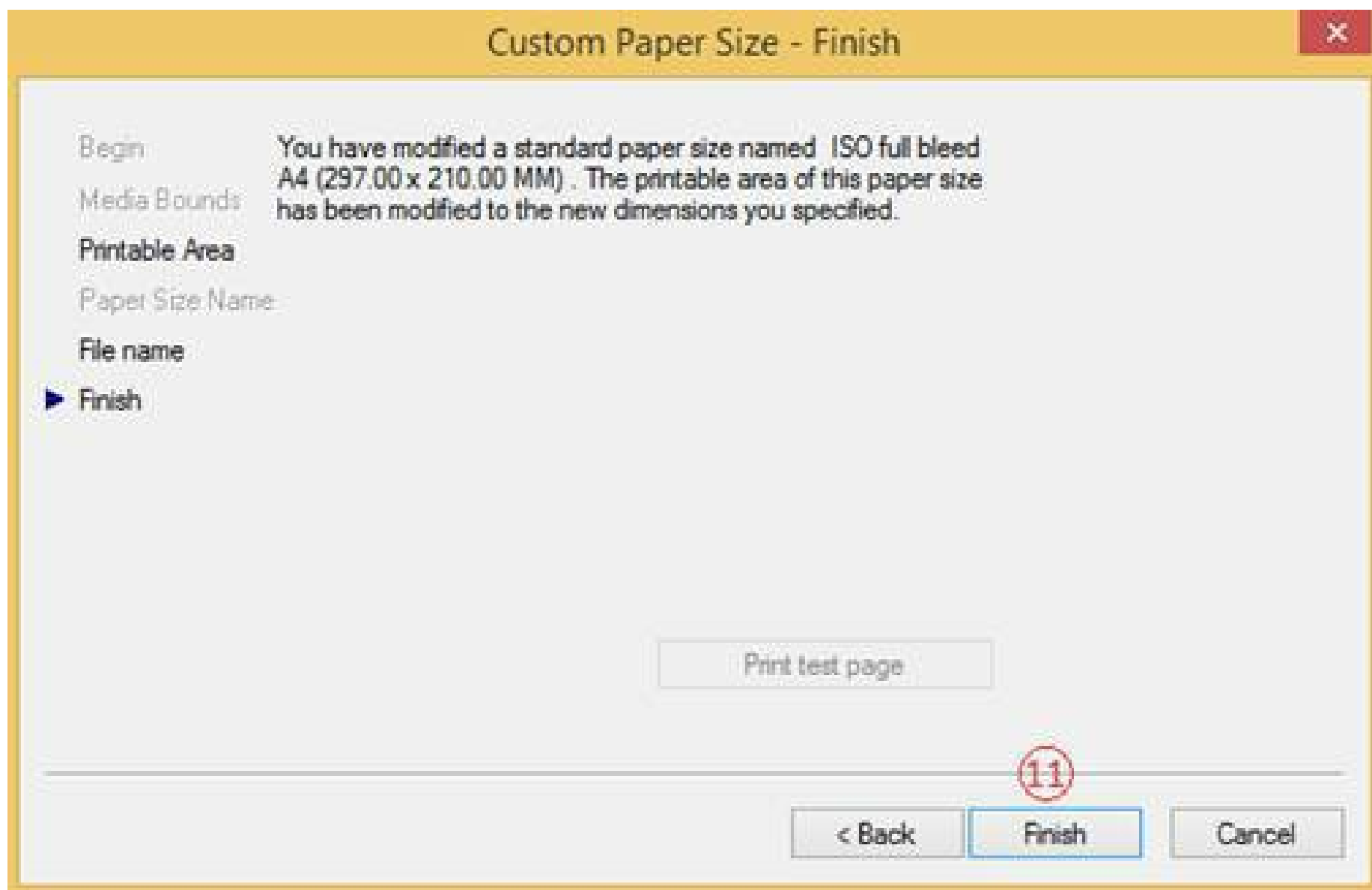
The new paper size will be stored in a PMP (Plotter Model Parameters) file. Enter a name for the PMP file you are saving.

PMP File name :

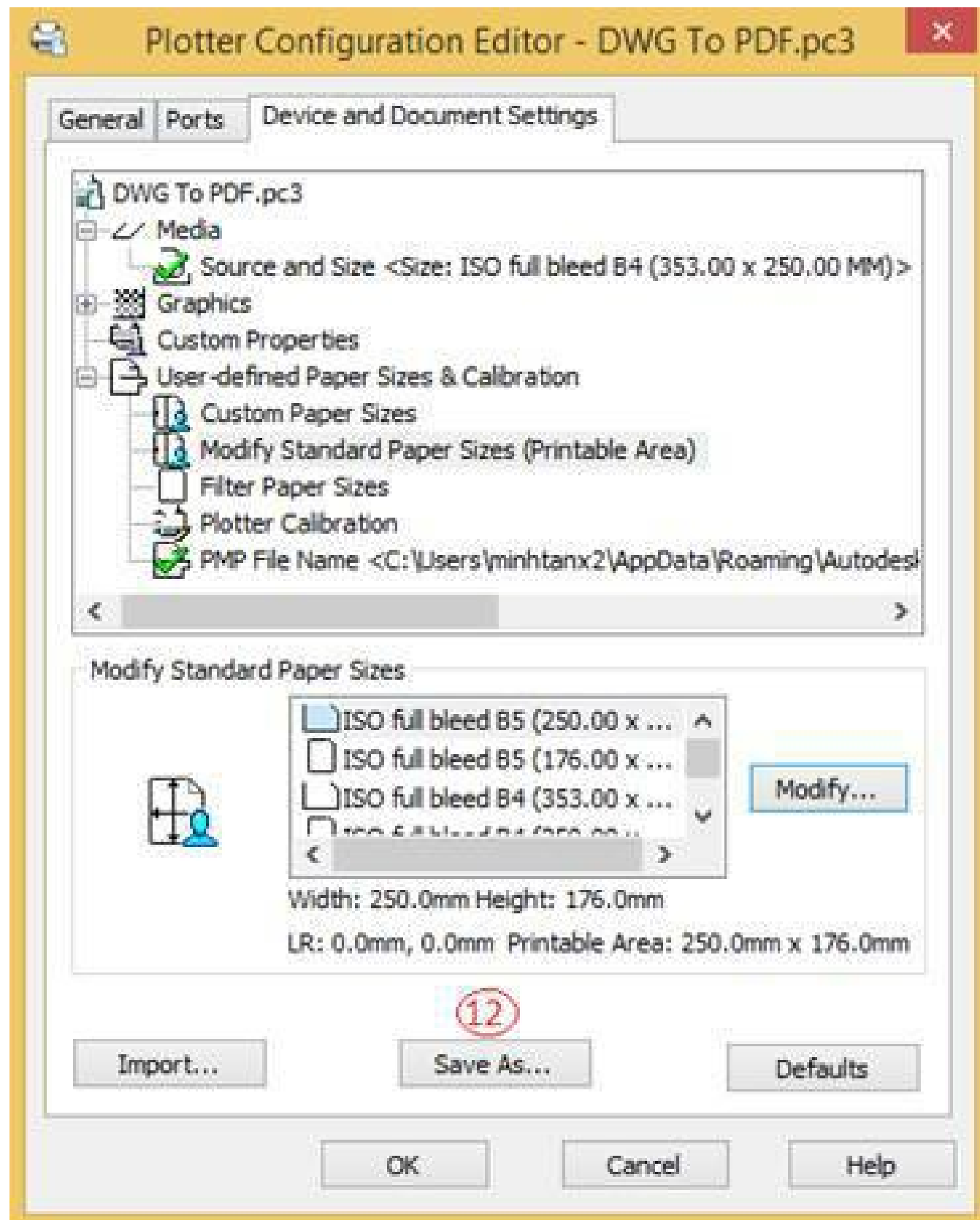
DWG To PDF A4 NGANG

10

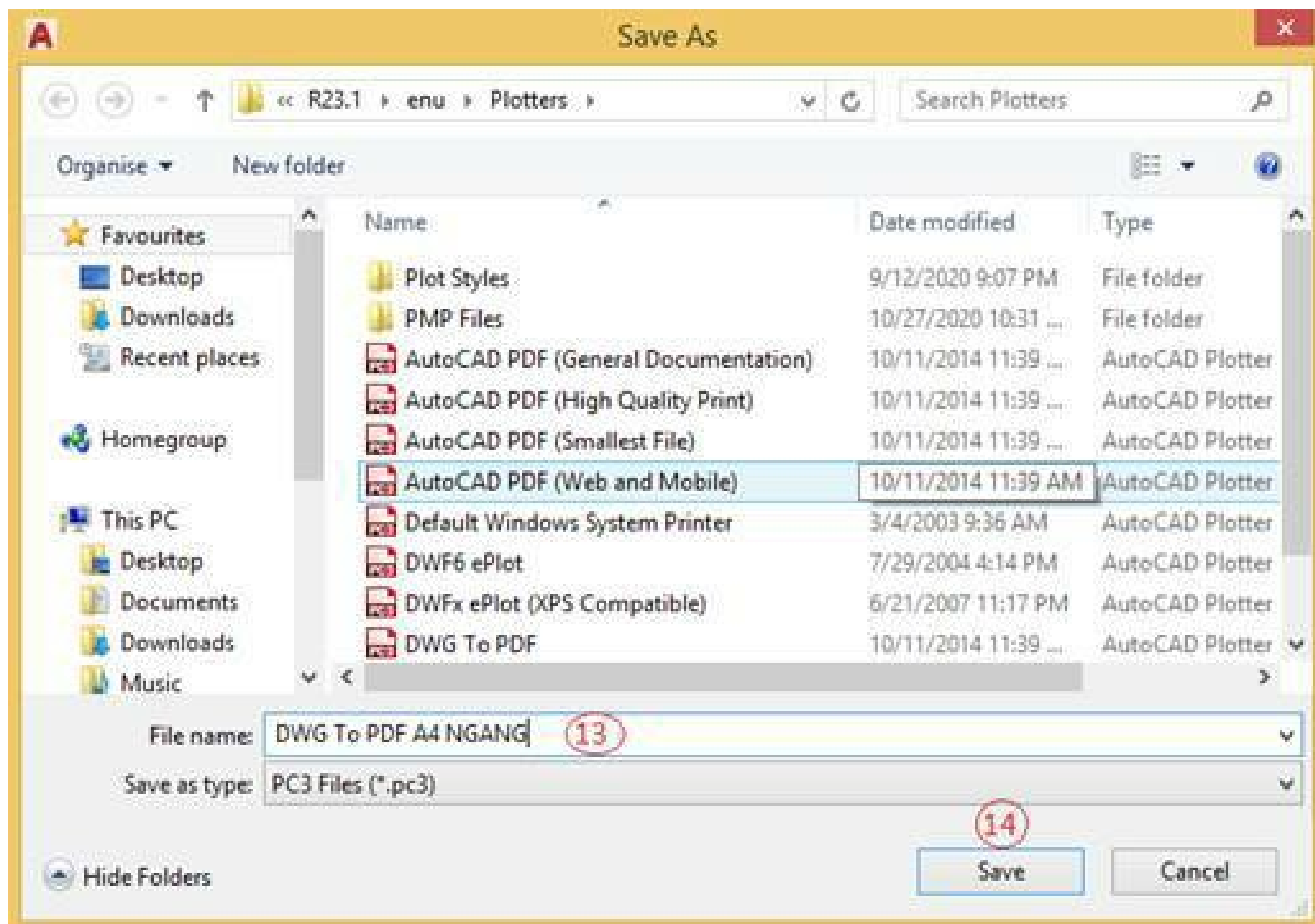
< Back
Next >
Cancel



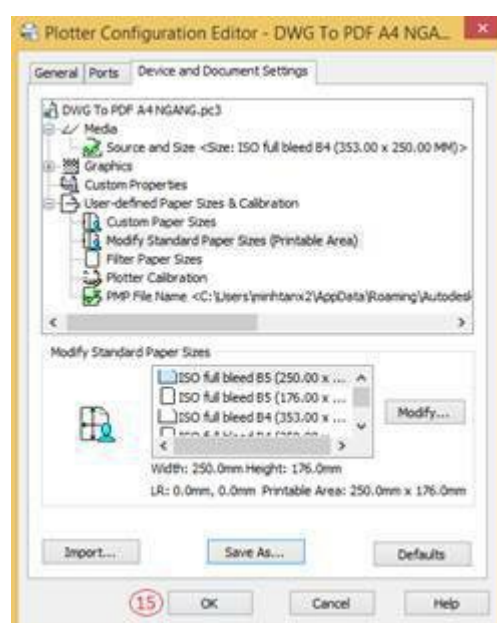
- Then click Save As... (12)



- Rename your printer model (13), then click **Save** (14) to save.



- Click **OK** (15)

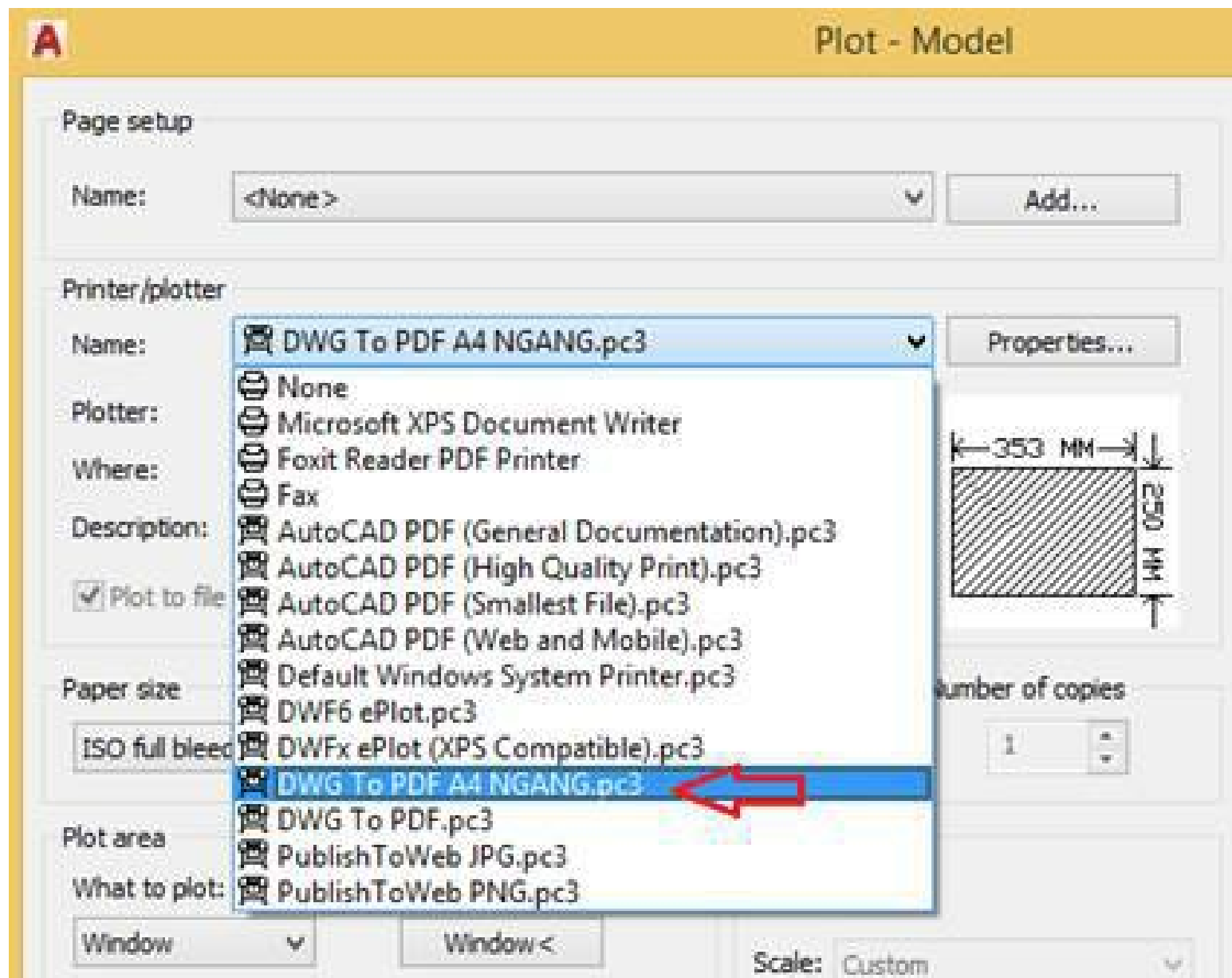


- Click on the item (16)



- Click **OK** (17)

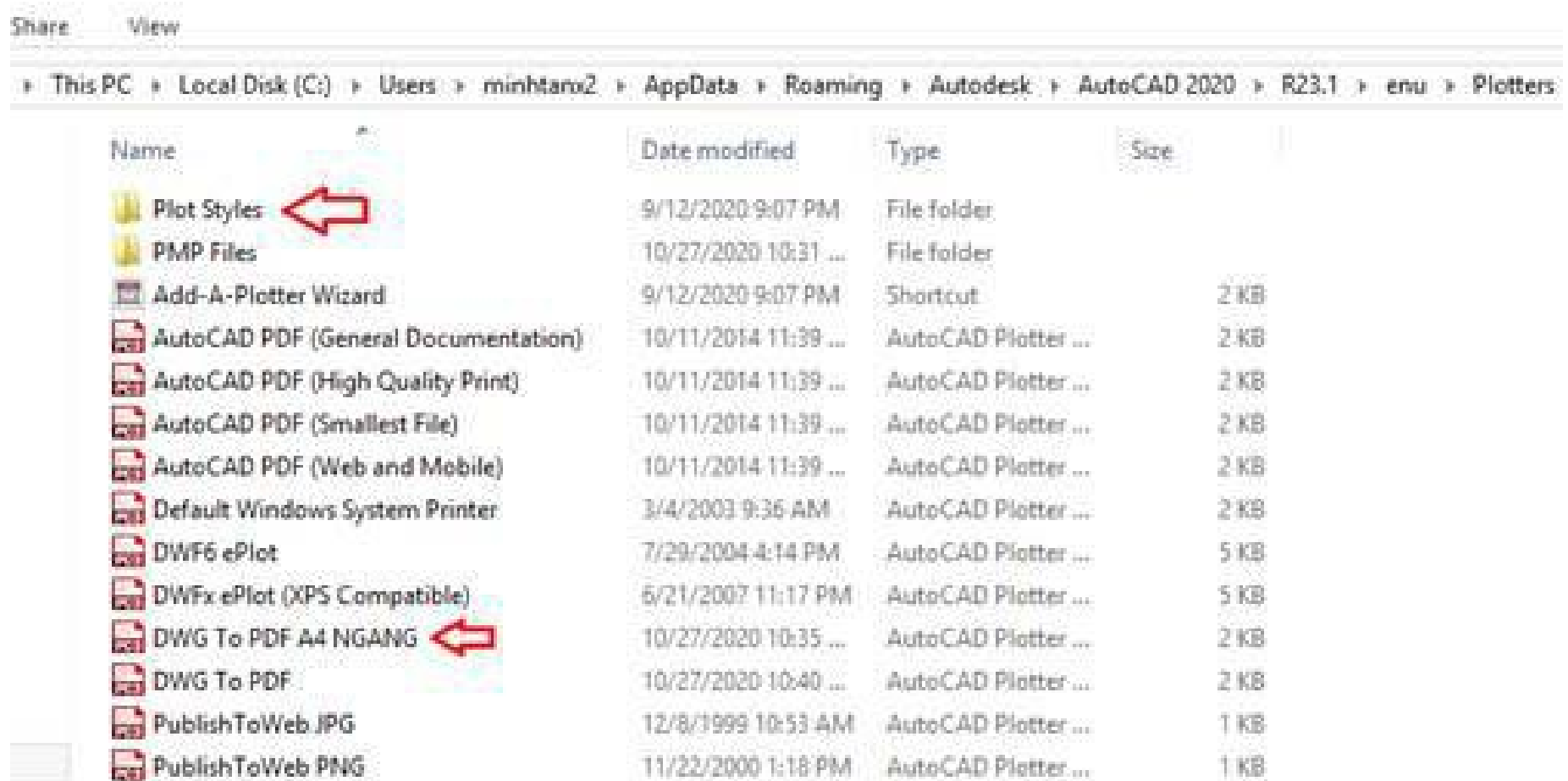
- Check in the list of printers, there is a new printer model you just created.



- With each type of paper size, you can create your printer model like that. From next time, each time you print to any paper size, real printer, or PDF, you will choose your printer model to create the corresponding. Make sure the correct rate is 100%.

Tell me more, the printer models you created yourself, where will they be located?

You just need to type the command **PLOTTERMANAGER**, it will immediately open for you the folder containing the list of available printers.



If you're interested click on the Plot Styles folder, there's a list of stroke types (including print color and stroke thickness).

Share View				
+ This PC + Local Disk (C:) + Users + minhtam2 + AppData + Roaming + Autodesk + AutoCAD 2020 + R23.1 + enu + Plotters + Plot Styles				
Name	Date modified	Type	Size	
acad	3/10/1999 4:17 AM	AutoCAD Color-d...	5 KB	
acad	3/10/1999 4:16 AM	AutoCAD Plot Styl...	1 KB	
Add-A-Plot Style Table Wizard	9/12/2020 9:07 PM	Shortcut	2 KB	
Autodesk-Color	11/22/2002 9:17 AM	AutoCAD Plot Styl...	1 KB	
Autodesk-MONO	11/22/2002 10:22 ...	AutoCAD Plot Styl...	1 KB	
DWF Virtual Pens	9/12/2001 3:04 PM	AutoCAD Color-d...	6 KB	
Fill Patterns	3/10/1999 4:16 AM	AutoCAD Color-d...	5 KB	
Grayscale	3/10/1999 4:16 AM	AutoCAD Color-d...	5 KB	
monochrome	3/10/1999 4:15 AM	AutoCAD Color-d...	5 KB	
monochrome	3/10/1999 4:15 AM	AutoCAD Plot Styl...	1 KB	
Screening 25%	3/10/1999 4:14 AM	AutoCAD Color-d...	5 KB	
Screening 50%	3/10/1999 4:14 AM	AutoCAD Color-d...	5 KB	
Screening 75%	3/10/1999 4:12 AM	AutoCAD Color-d...	5 KB	
Screening 100%	3/10/1999 4:17 AM	AutoCAD Color-d...	5 KB	

We can completely create our custom print styles, but I find it unnecessary because we are printing in layers. Installed in layers, no need to think when printing. Print is print, that's all!

I just showed you the basics of printing a drawing in AutoCAD's Model space.

Did you find anything new, let's be honest!

2. Mass print AutoCAD drawings on Model

Now let's move on to more advanced knowledge. It is mass printing. There are many different types of mass printing, be it printing on Model or Layout.

However, printing on the Layout does not seem to be very suitable for the mechanical characteristics. Therefore, I will only introduce you to the method of mass printing on the Model, most simply and effortlessly.

How to do if you want to print mass drawings on Model?

The fastest way is to use AutoLISP (referred to as Lisp)

Have you heard of AutoLISP? In a nutshell, it is a form of control code that is programmed from the outside and inserted into AutoCAD so that it performs a certain complex request that AutoCAD cannot do by default. Its format extension is usually .lsp or .vlx (visual LISP). For .lsp, you can edit freely, while .vlx is not.

I know there are a few lisps that can do Model side bulk printing, however, I found the “TPL-DNM” lisp to be suitable and I recommend it to you. Your job now is to insert it into AutoCAD how to turn it into a permanent command in AutoCAD such as the Line command, Rectang, Block, ... only. That's it!

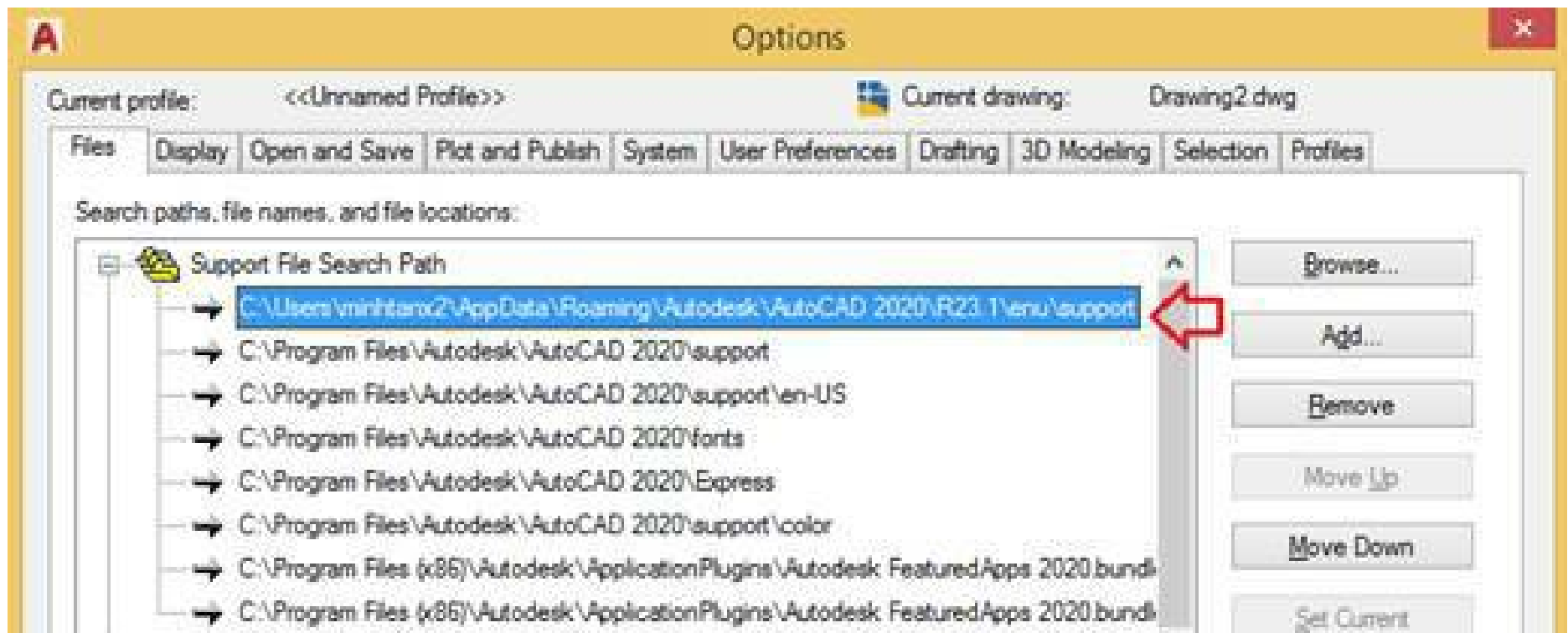
Keep going!

Do exactly what I told you to do!.

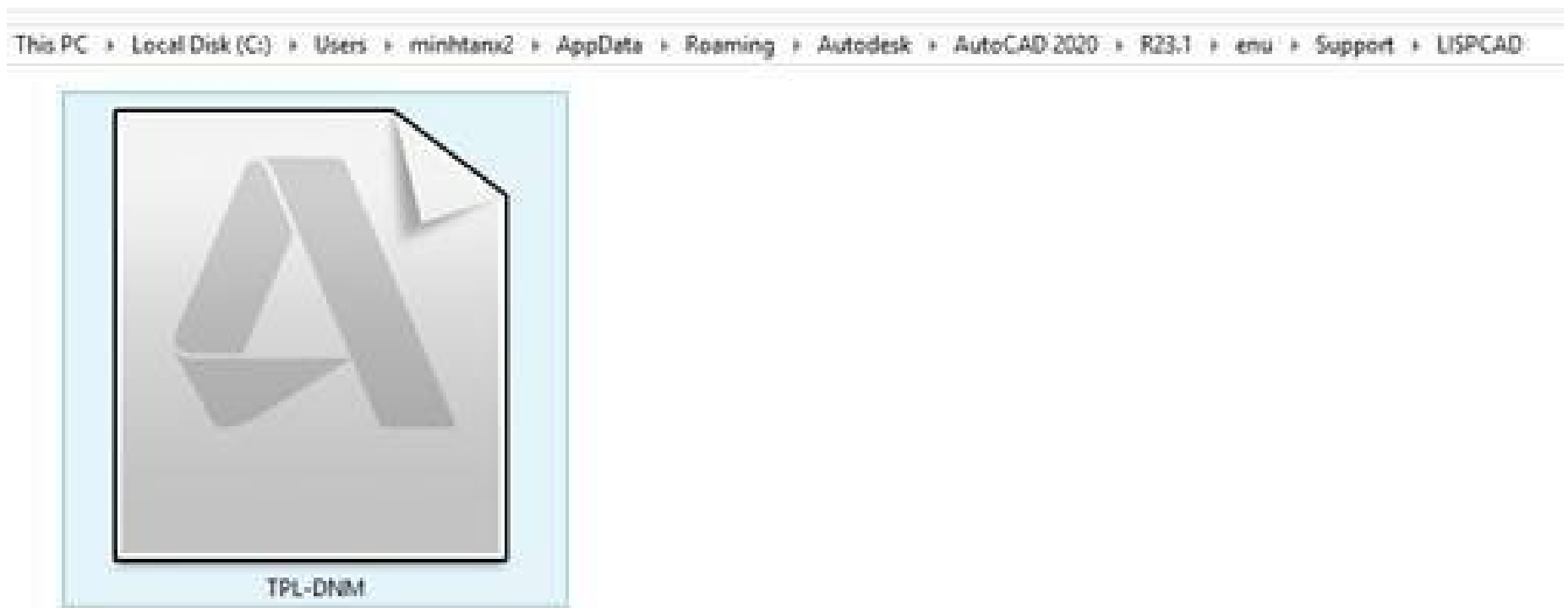
2.1. Insert AutoLisp into AutoCAD

Link to download the mass printing lisp: <https://tinyurl.com/h23umkc>

- Create a folder named LISPCAD inside AutoCAD's Support folder. Depending on each AutoCAD version, the path to the Support folder will be different, you can see the exact path on your computer here:



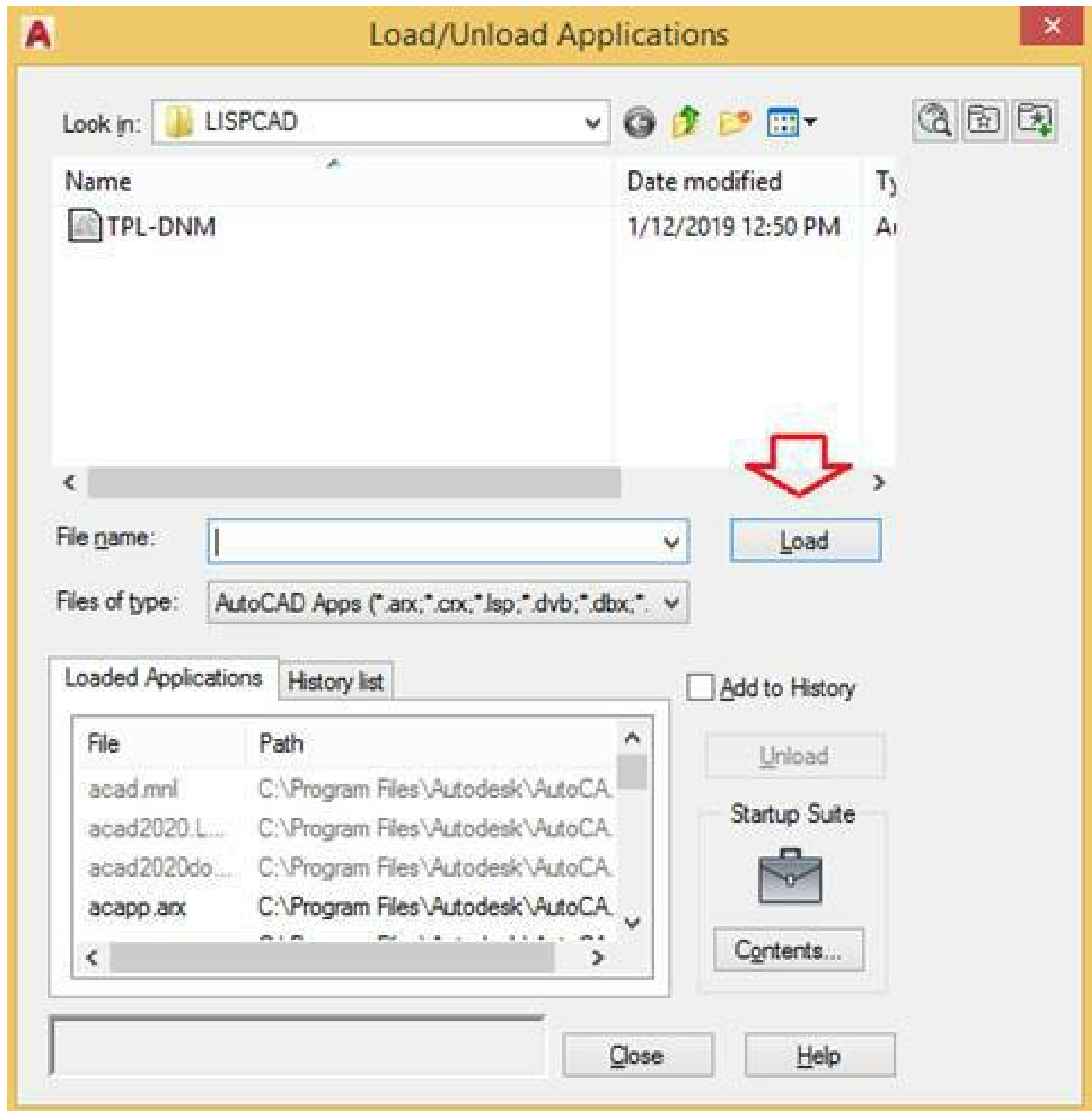
- Copy the mass print lisp and put it in the newly created LISPCAD folder



The purpose of doing this is to put the Lisp file directly into AutoCAD's startup system. If not included here, every time AutoCAD starts, it will probably ask you “Do you want to reload this Lisp?” which will be annoying every time you open AutoCAD. By the way above, I have helped you to prevent that annoying thing right away.

- To insert Lisp into the system, type the command: **AP** (Appload)

In the Look in section, find the path to the LISPCAD folder in the Support folder you just created



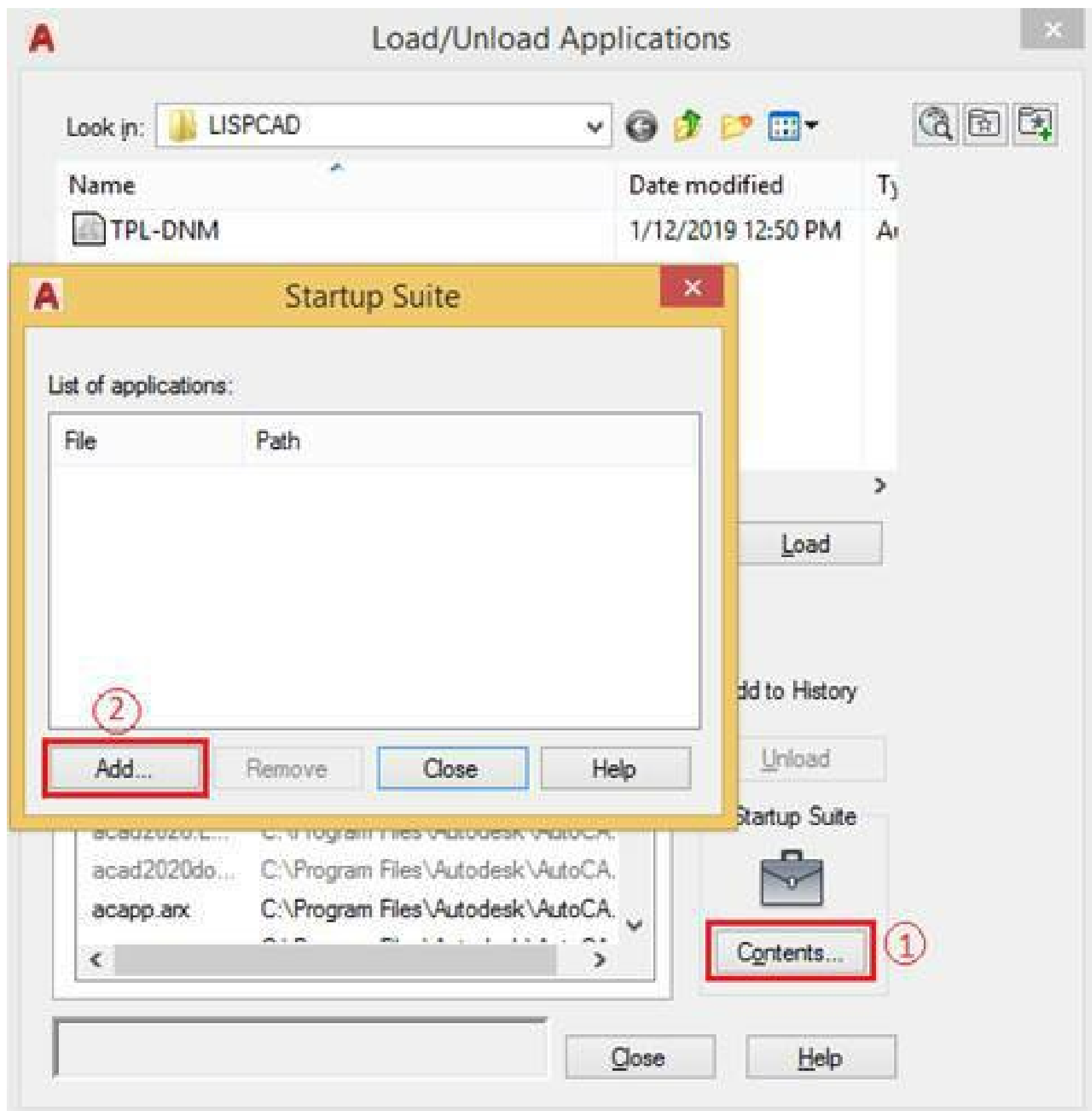
- Select the lisp files “TPL-DNM”, then click “Load” to activate them into the current drawing.

--> At this point, we can use the TPL command of AutoLISP to mass print.

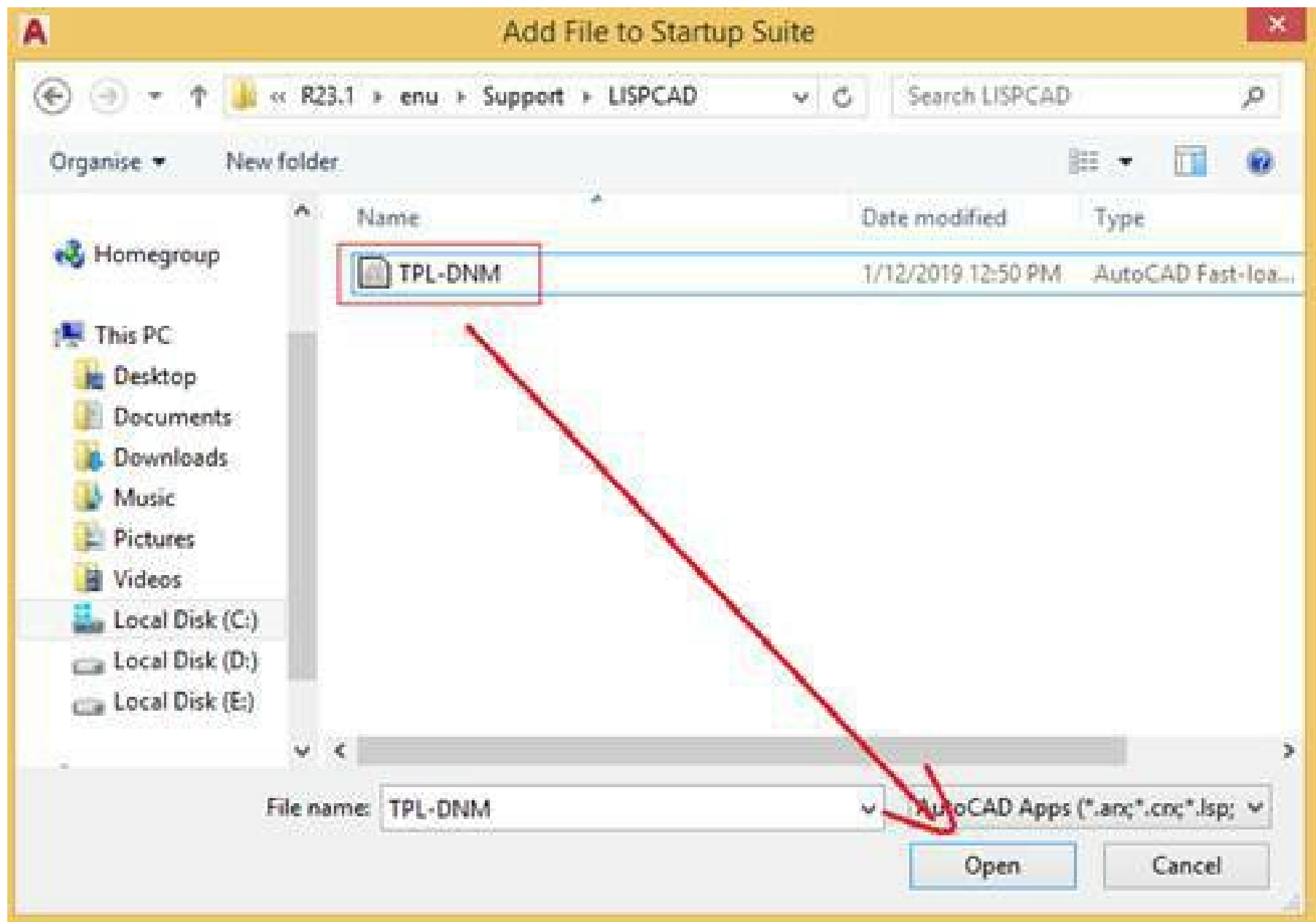
Try Close the dialog box again and type the TPL command ✓ see what it outputs?

However, if you open another drawing or restart AutoCAD, this Lisp still cannot be used.

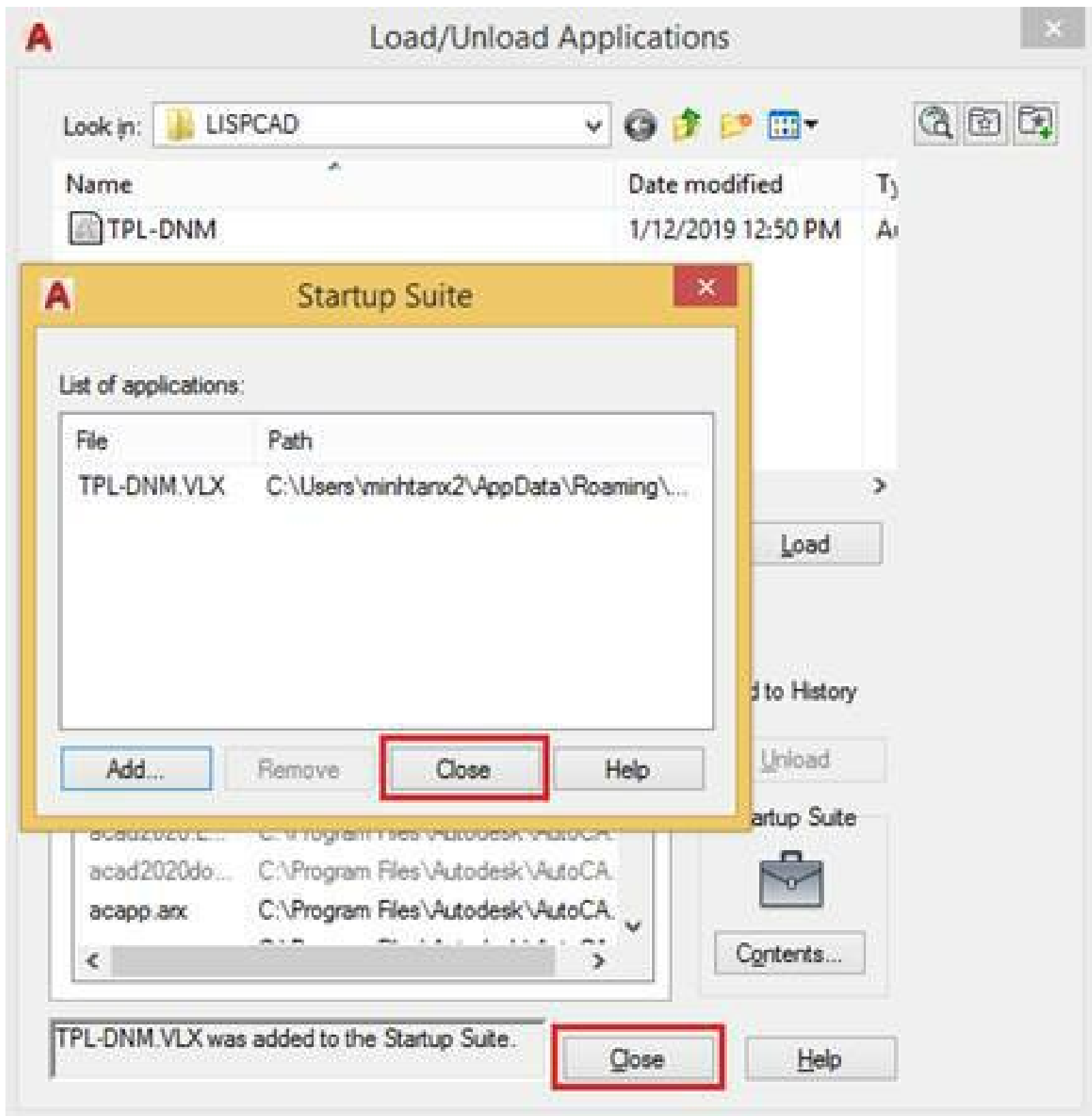
To be able to use this Lisp as a permanent AutoCAD command, still in the Appload command dialog: click Contents, then select Add



- Click again to select the lisp file "TPL-DNM", then select Open



- Click Close/Close (2 times) to close the Appload dialog box.



Now you can use this Lisp carefree.

If you have other Lisps, you can also throw them in here and do the same.

2.2. How to mass print AutoCAD drawings

I will tell you exactly how I am doing. And your job is just to follow, but if you have a better idea or idea, even better. I am only sharing my experiences, sharing what I think is the best, most effective, but I also would love to hear your new ideas. I want to learn from you too! You are so smart, please share with me your cool ideas!!!

To mass print, we will need to go through some steps, you can rest assured that it is not complicated, not complicated at all, just as simple as walking.

Step 1: Arrange all the drawings you want to mass print in the same file

I think you are already doing this because we manage drawings by the project. All drawings of each project should be placed in one file.

Step 2: Use the PRINT layer to redraw the print frame border for each drawing

Do you remember in a chapter on layer management, I mentioned that I needed to create a specialized layer for mass printing, I named it the PRINT layer. Do you remember? If you want to order something else, that's okay too.

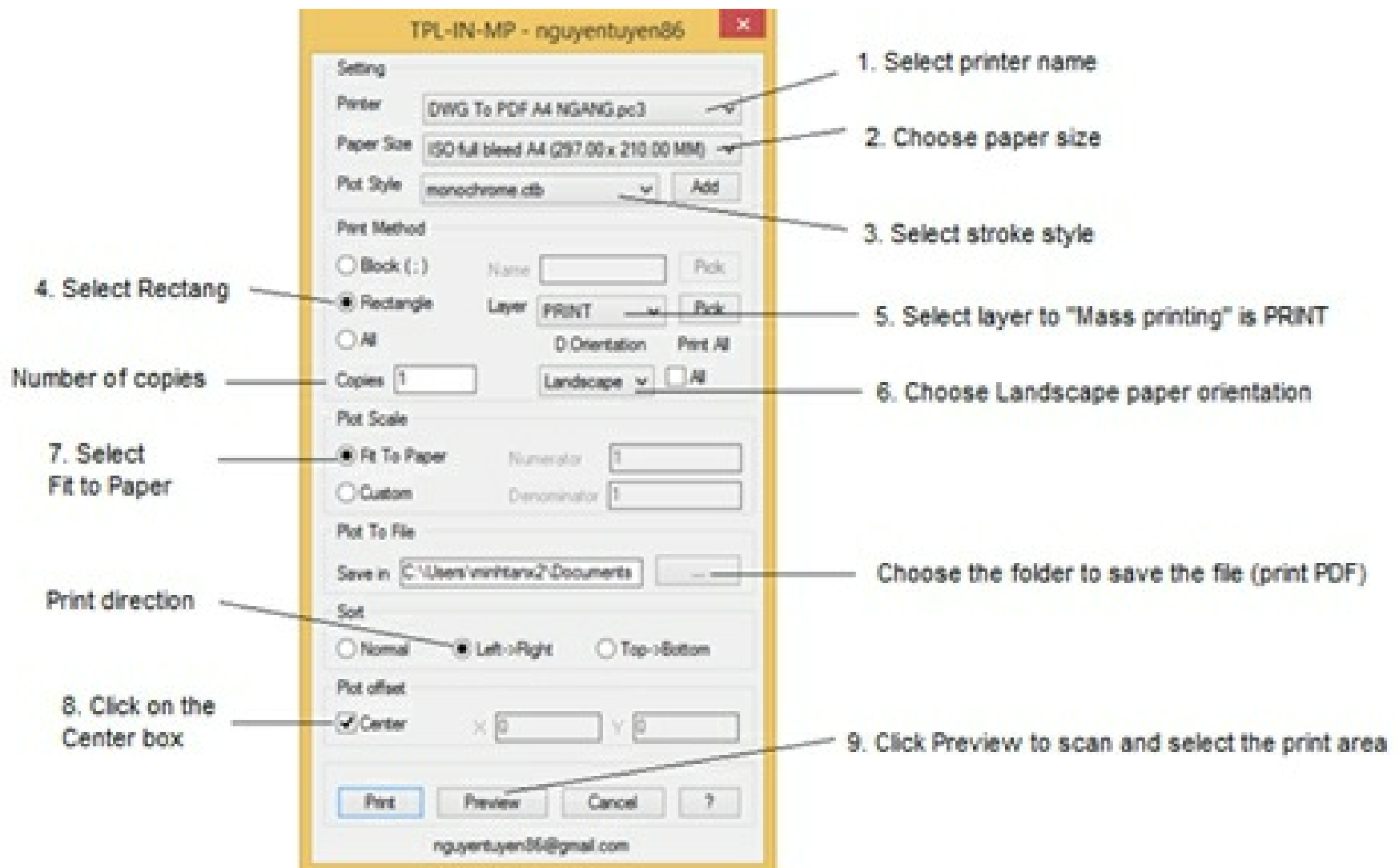
This PRINT layer does not need to be sophisticated, because it is not printed, but just a border around the drawing frame for Lisp to print in bulk to recognize the print area inside. You should let the color of this PRINT layer match the color of the name frame border (I chose cyan) to avoid exposure.

Every time you finish a drawing, your job is just to use the RECTANG command, draw a rectangle around the name frame of that drawing, and that's it. Remember that this rectangle must belong to the PRINT layer!

Step 3: Conduct mass printing.

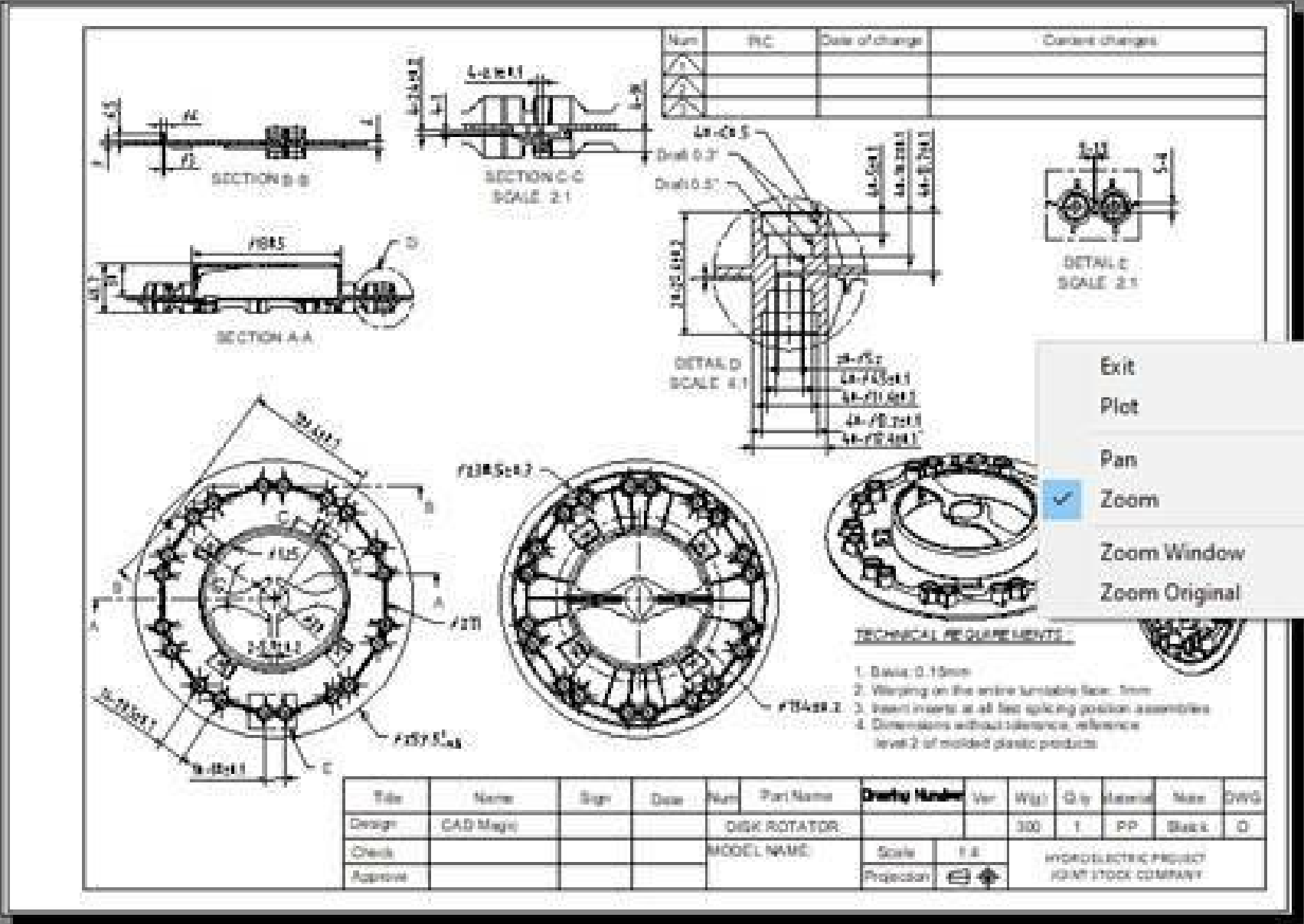
That's all you need to prepare to print in bulk.

- Type the command TPL ✓ to call up the command's dialog box.
- Set steps from 1 to 9 as follows:



Don't worry, only the first time you have to install many things, then it will save your settings automatically. From the second time, you just select the drawing object to print again. It doesn't take much effort.

- After you click **Preview**, scan through the entire area containing the drawings you plan to print. Feel free to scan, it will automatically capture any drawing that has a drawing frame created by the PRINT layer.
- To remove (not print) any drawing, you just hold Shift and click on the border of that name frame to "deselect"
- After you have selected all the drawings to be printed, press the Enter (or Space) key to finish selecting the drawing.
- The first drawing appears (Preview). You right-click and select Plot and it starts printing.



- The second drawing appears. You right-click and select Plot and it is serialized

- The third drawing appears. You right-click and select Plot and it is serialized

...

And so on until all the drawings are gone.

If there are 100 drawings then you will have to right-click and select Plot 100 times.

If you print in Layout, you will only right-click and select the correct Plot once for hundreds or thousands of drawings. But this is the Model side and that is also the limitation of Lisp. But I know for sure it's still much faster than having to do it manually, scan and order 100 times for those 100 drawings. The more drawings you have, the more clearly you will see the time difference between Lisp and no Lisp.

If you want to cancel any drawing, just right-click and select Exit. It will automatically skip that drawing and immediately go to Preview the next drawing.

Let's practice now boys!

How do you feel?

.....
.....
.....

You've just been equipped with a new weapon, haven't you?

Can you share a little bit about your feelings about the book, so that I have more motivation to be ready to continue?

Chapter 18: Create Your Template File

As I said, a Template is indispensable for those who use AutoCAD software. The template contains all the settings we have set up, including drawing unit, Layer, Linetype, Text style, Dimstyle, MleaderStyle, Block, Name frame, Text, ...

When setting up a drawing for a new project, you just need to use the Template file and you already have all the tools in it. It contains all the things that you need to use often. It's like just throwing all your gear in a bag and taking it anywhere on Earth. Templates help you always be ready to fight in any situation. You will be a professional fighter.

Template to me is an indispensable toolkit, the luggage I always carry with me. Such sophisticated items, literally only for high-class characters like James Bond.

In Bear Grylls' "Survival Skills" series (I'm a big fan of this guy), in all the most extreme situations: whether it's desert, volcano, jungle, glacier or deserted island, ... Bear Grylls always carries with him 3 "unreasonable" objects: 1 water bottle, 1 knife, and 1 ignition device.

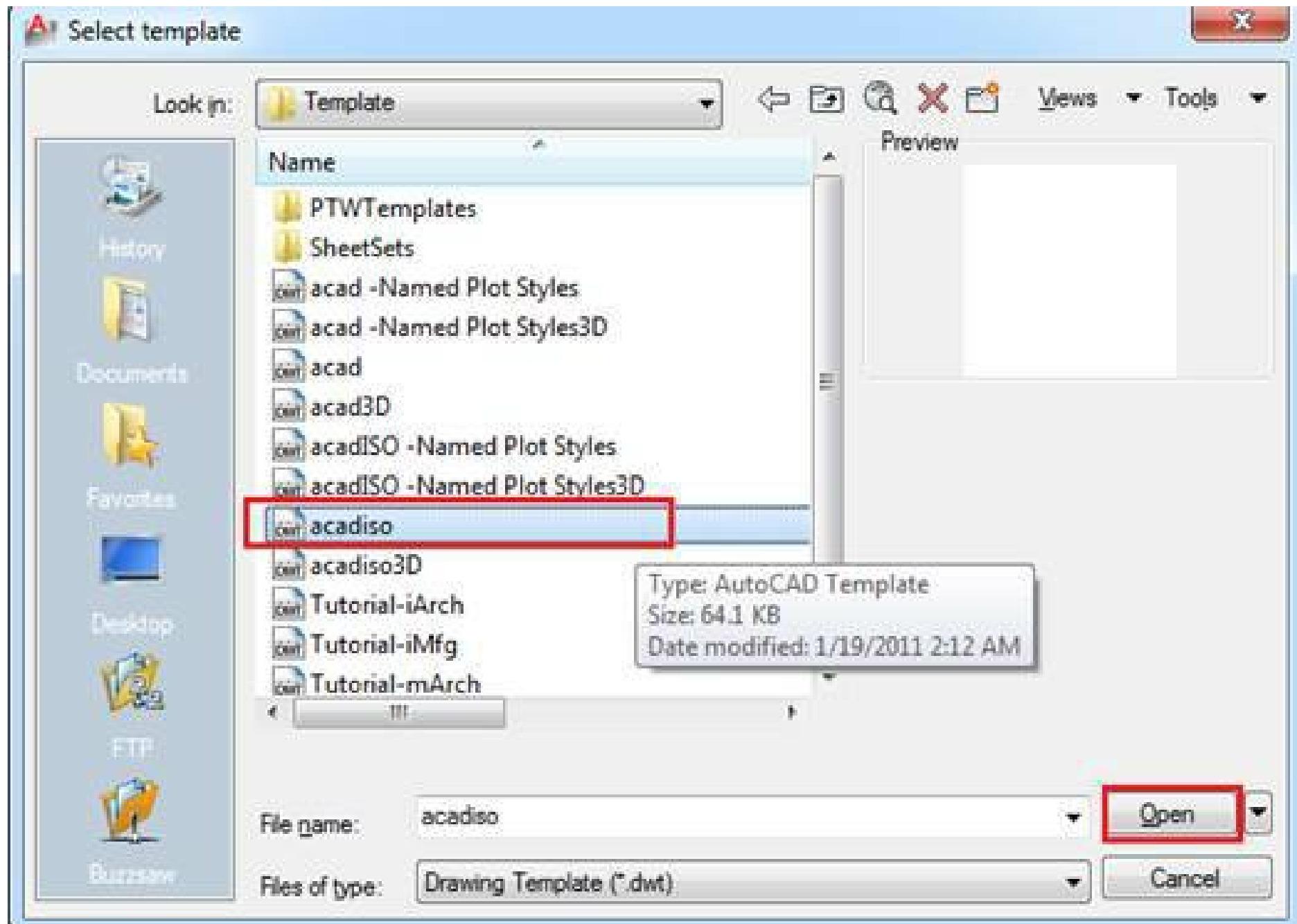
And you will also have much luggage, but with AutoCAD, you only need 1 Template and 1 shortcut file to be able to step on the stage!

Start your journey now!

1. How to create a Template file

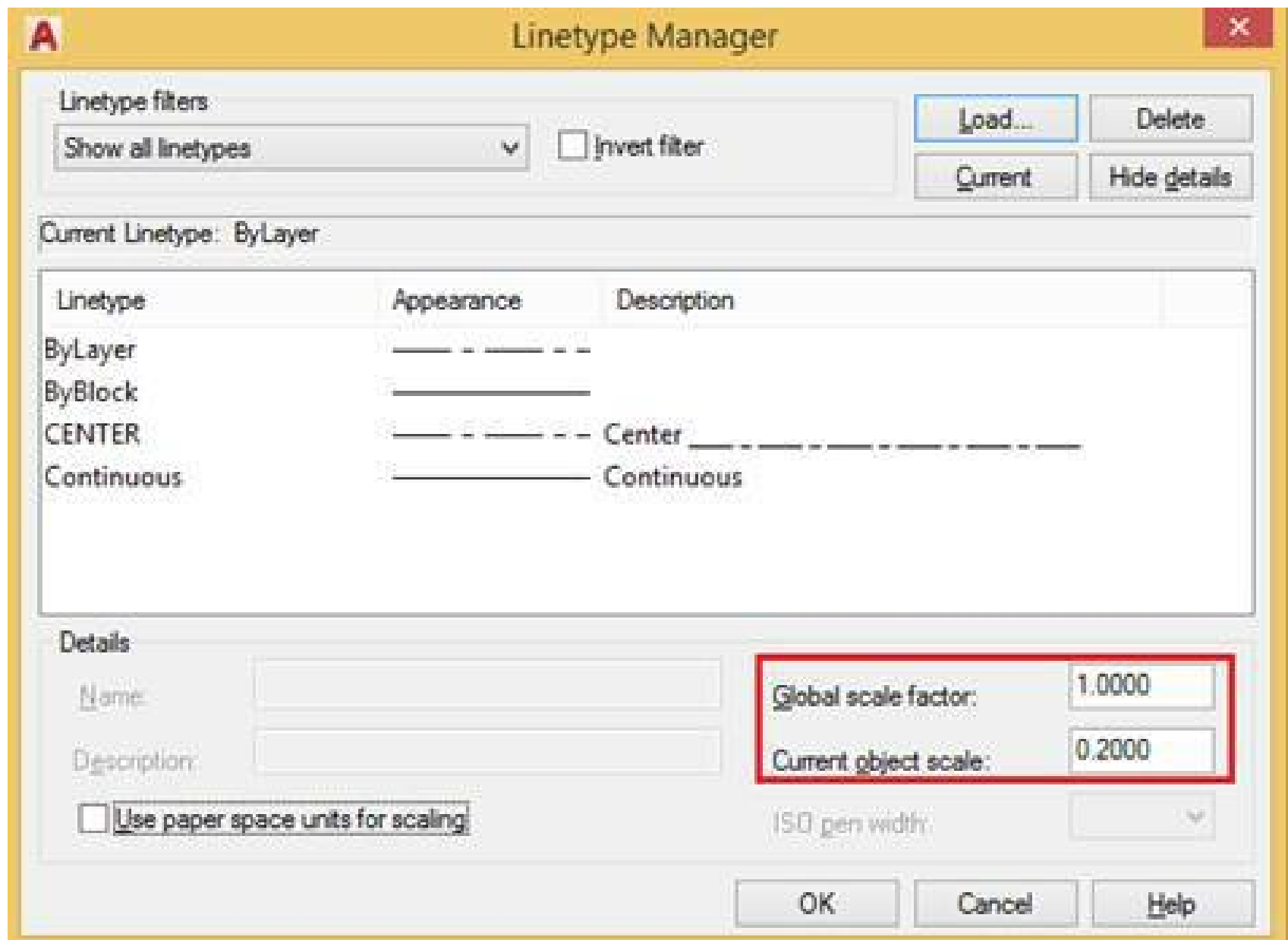
To create a Template drawing file, the process is as follows:

- Press Ctrl N to open a new drawing file
- Choose any Template file to open, for example, I choose acadiso.dwt
- Open



- In this new drawing file, we set the settings about:

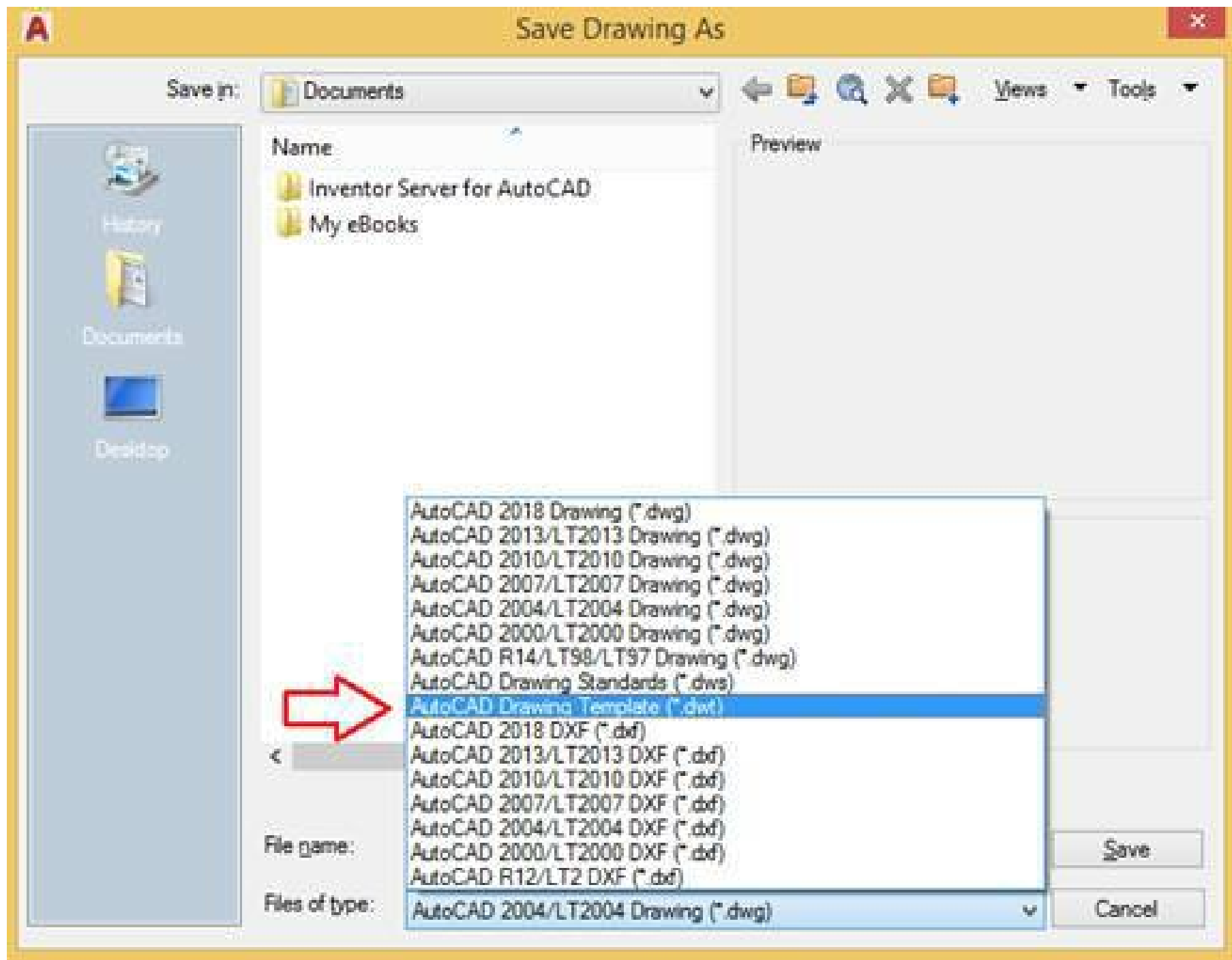
- Set drawing unit (mm)
- Layer: set layers according to the company's regulations on lines, colors, ...
- Textstyle: set style "Textnote 2.5", "textdim 2.5" (Textnote 3.5 and textdim 3.5 if A3 size)
- Dimstyle: set dimstyle "DIM 1-1"
- Mleaderstyle: style "Textnote 2.5"
- Set dashed line scale for DRAWING SCALE 1:1 in Linetype Manager dialog box



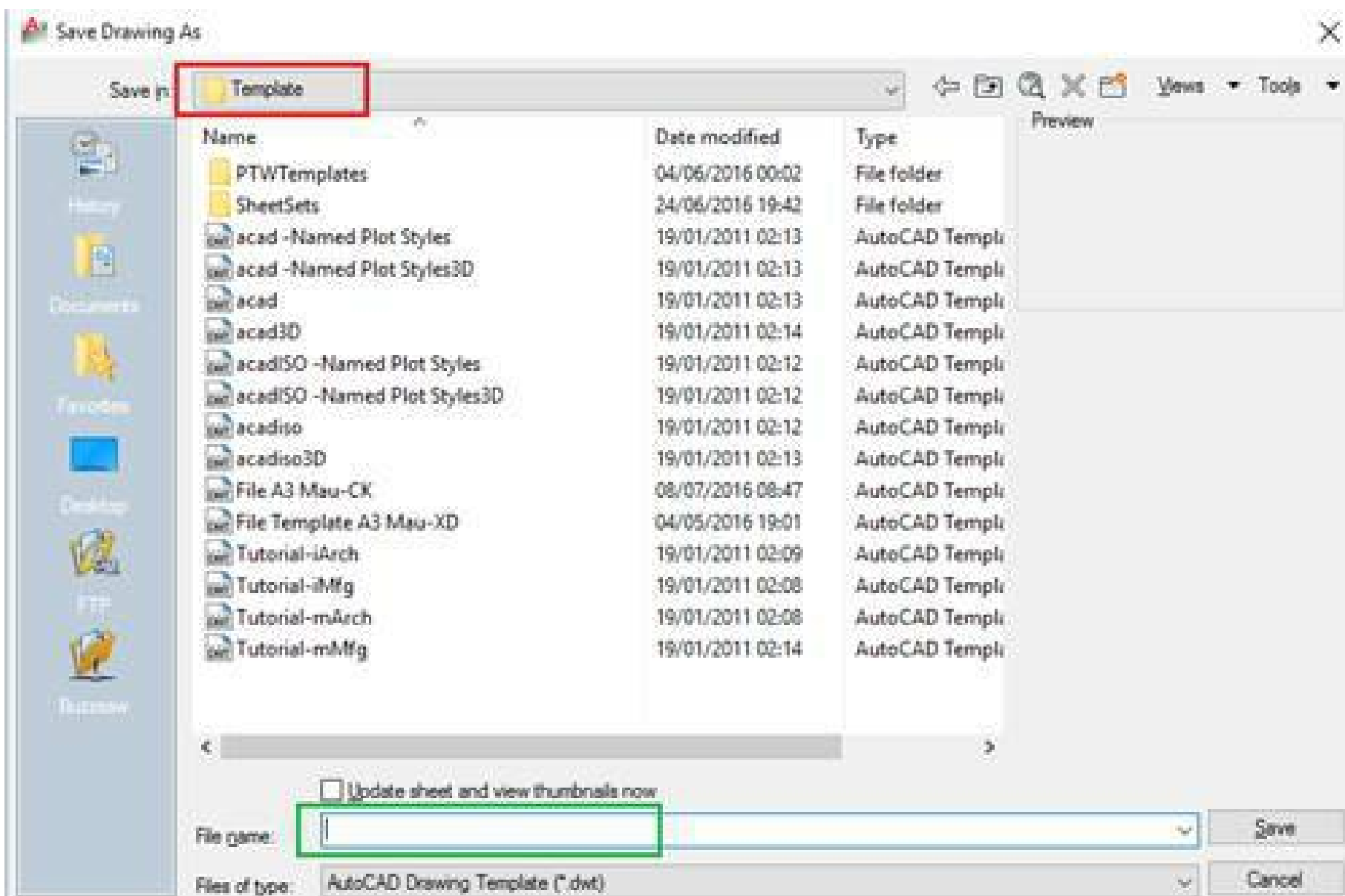
- You can insert Blocks, then delete them from the screen, but it is still saved in the Block library of the file
- You can add a title block
-

If you want to have any settings available, set them upright in this file to include in the Template file.

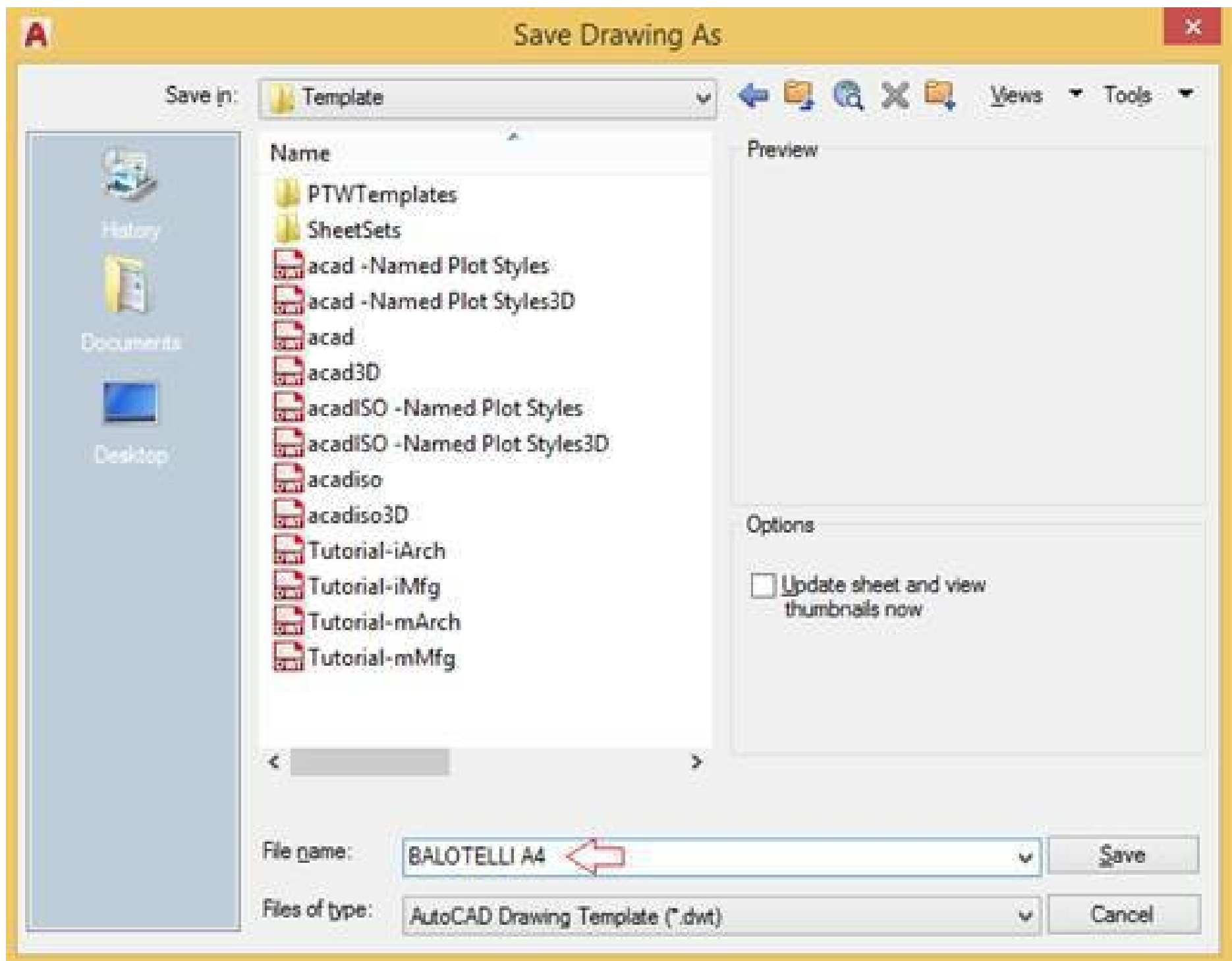
- Install Options: the Options setting will not be saved in the Template, but you should always install it in this file to practice meticulousness and one more memory.
 - Set Draft Setting, ...
- Press Ctrl S to save the drawing, in the Files of Type section select AutoCad Drawing Template format (*.dwt)



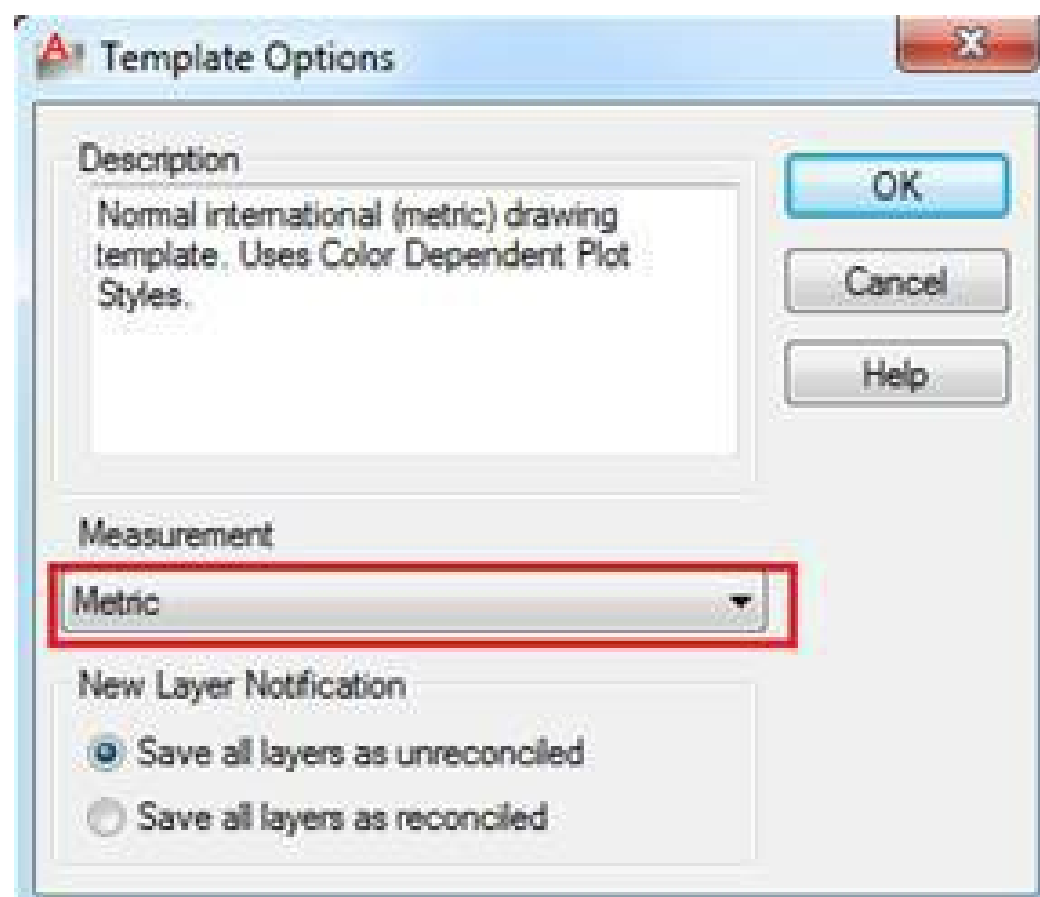
- Name the Template file in the File name box



For example, here I name the Template file: BALOTELLI A4.dwt



- Save/ Select Metric unit/ OK



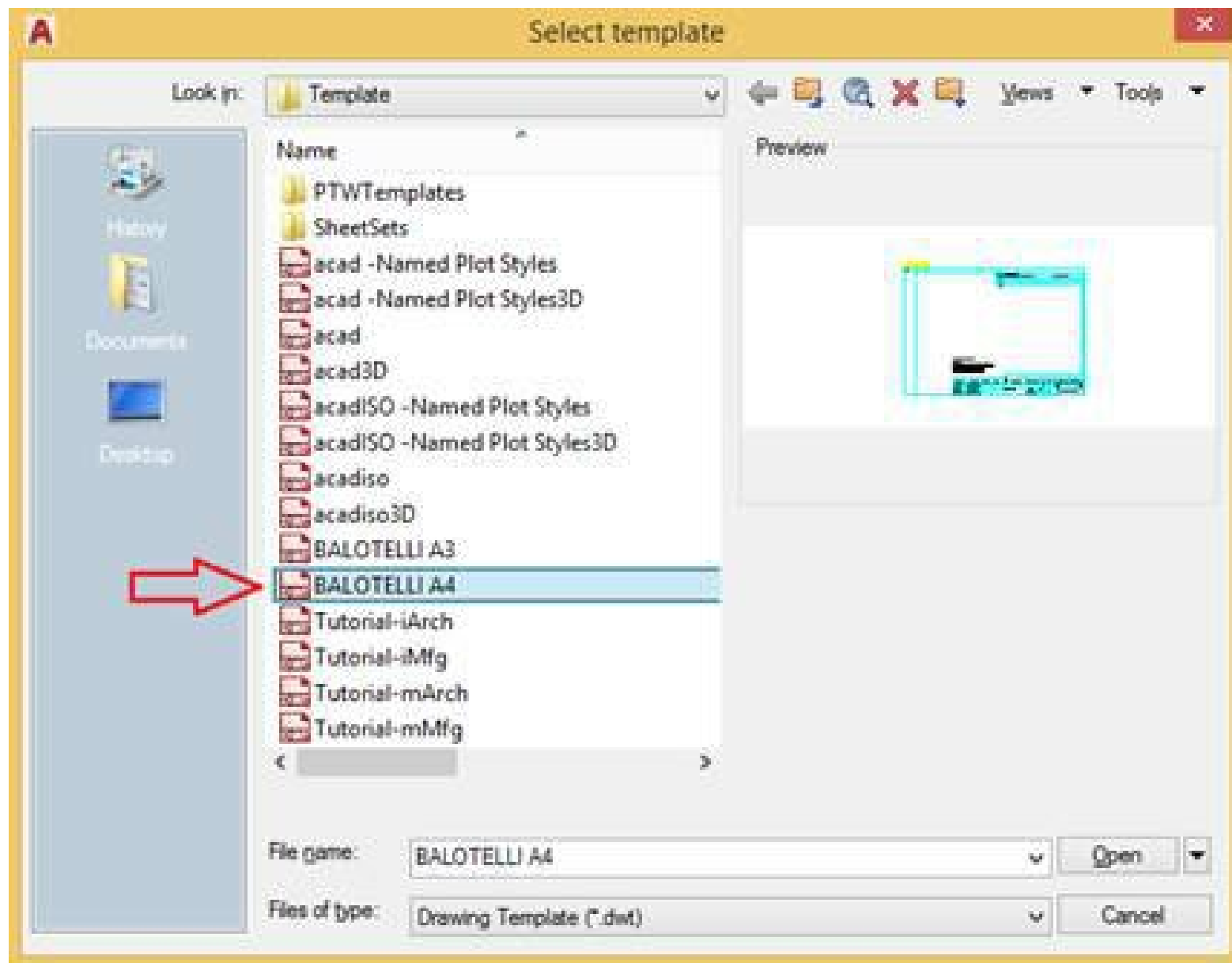
So we have already created a sample drawing file for the Model space for AutoCAD.

2. Using Template files

2.1. Create a working drawing file every time you open AutoCAD

To create a new drawing file to work with, do the following:

- I press Ctrl N
- Select the template file previously created (eg: BALOTELLI A4)
- Open



Now, we have a new drawing file (.dwg) containing all the available settings to get you ready to go.

2.2. By default, only open your Template file every time you start AutoCAD.

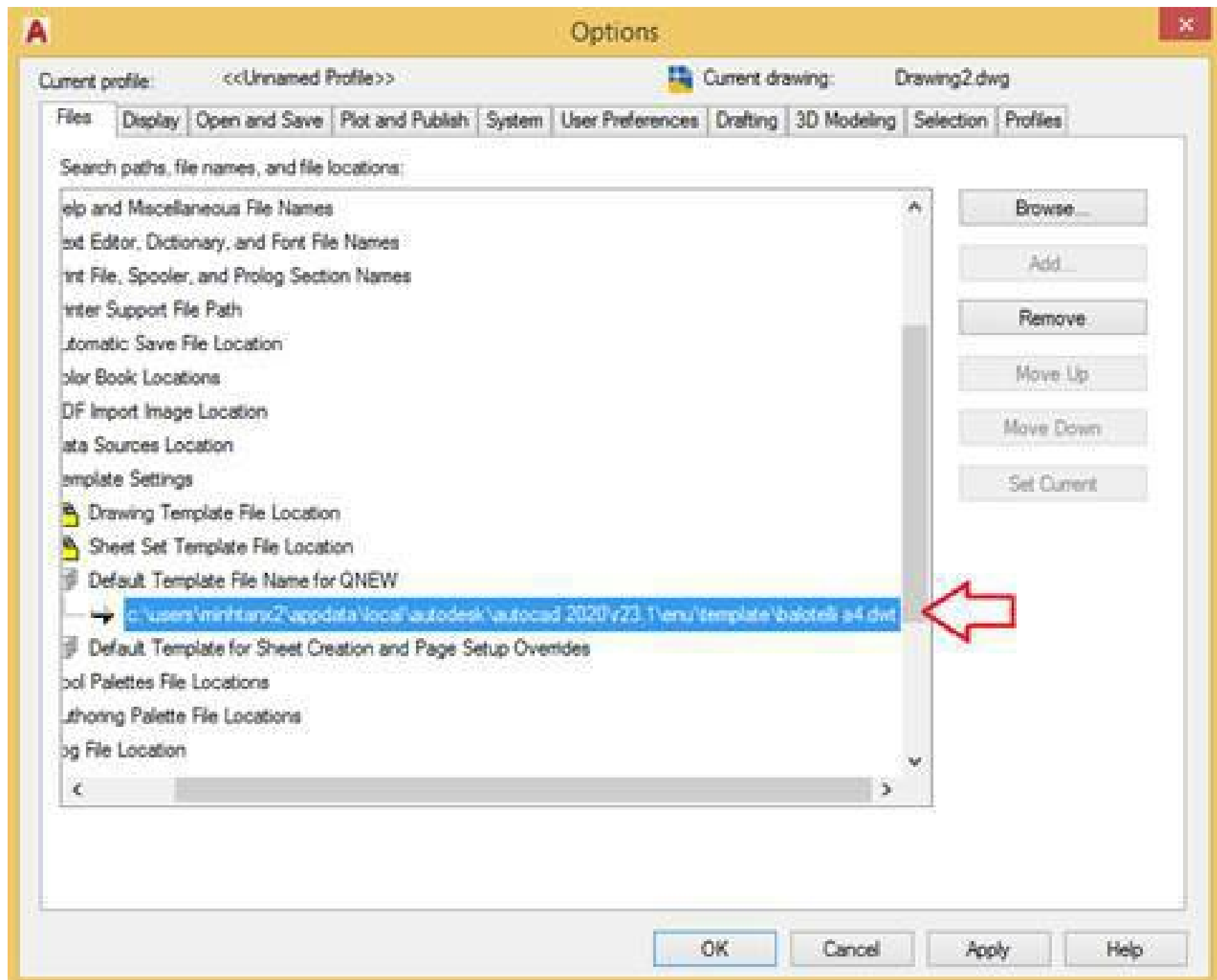
Question:

If I want every time I turn on AutoCAD software, the machine will automatically open the main new file from my Template file, is it possible?

Okay! You do the following:

Step 1: Set variable START UP = 0

Step 2: Go to *Options / Files / Template Settings / Default Template File Name for QNEW / Browse /* Select the path to the Template file we want to open by default at software startup / Open / OK.



Here, every time I run AutoCAD software, it will always open a new file for me from the Template file that I created before, BALOTELLI A4.

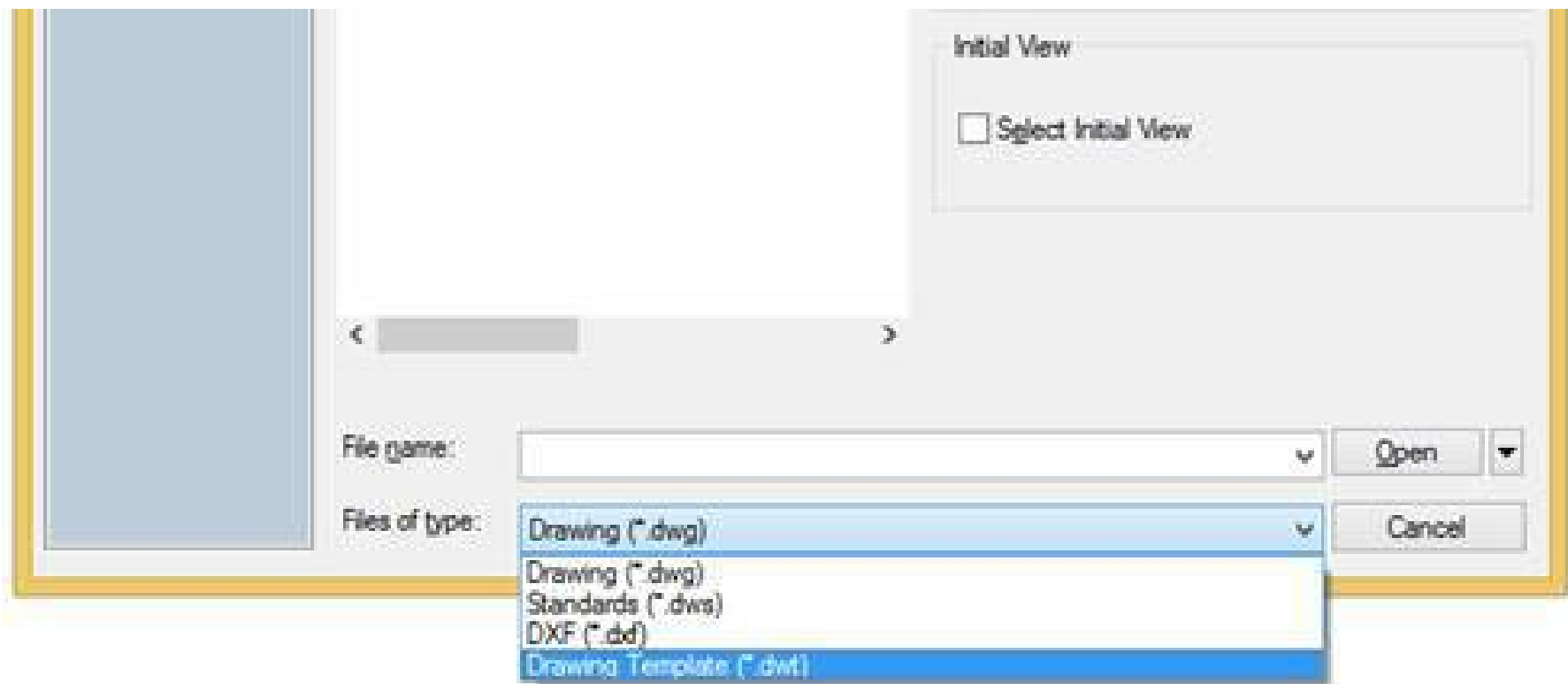
And you will do the same!

3. How to edit the Template file

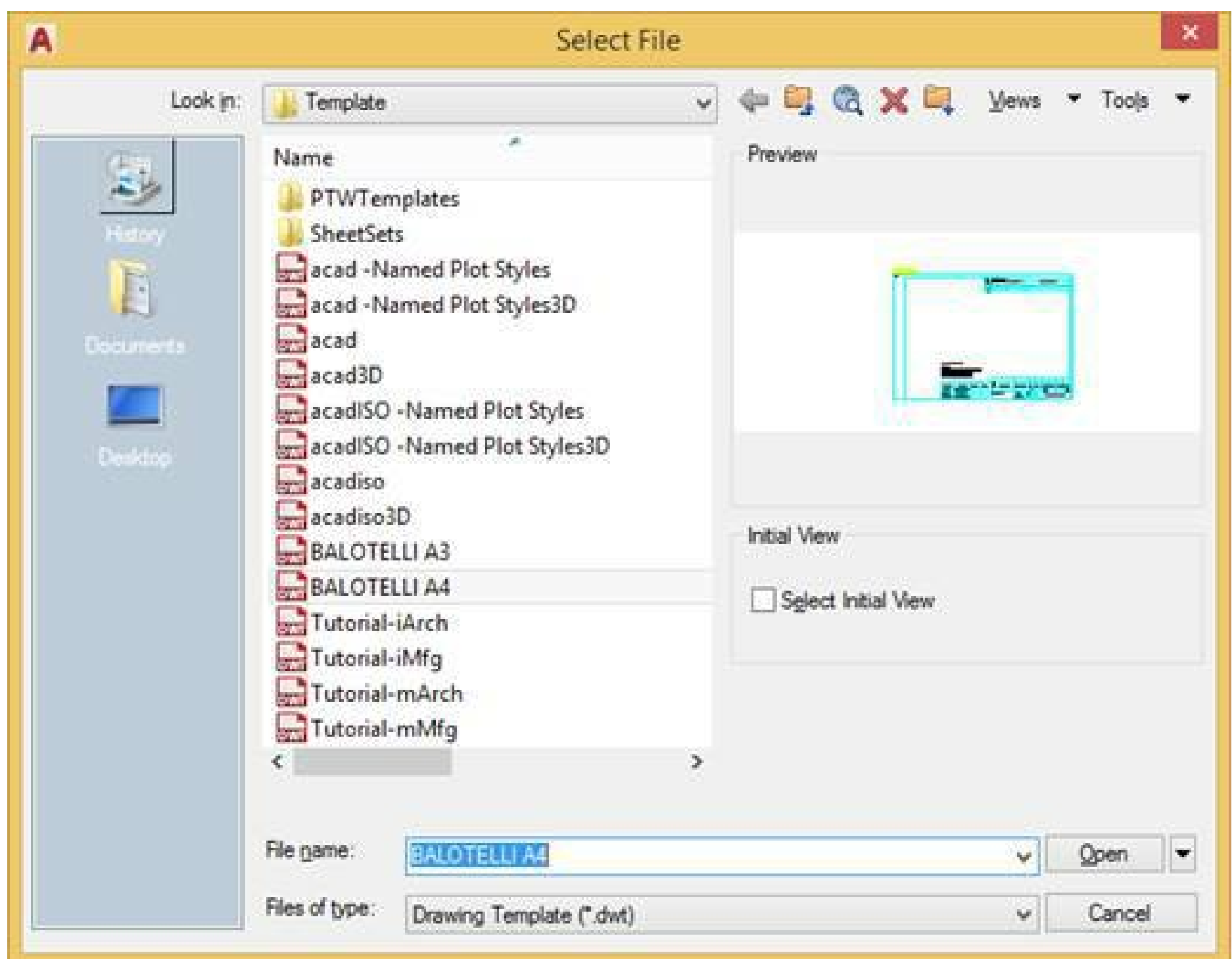
What if I need to modify the Template file? Is it possible to reset from the beginning another Template file like that?

No, we can reopen that Template file and edit it as we please. Then save it.

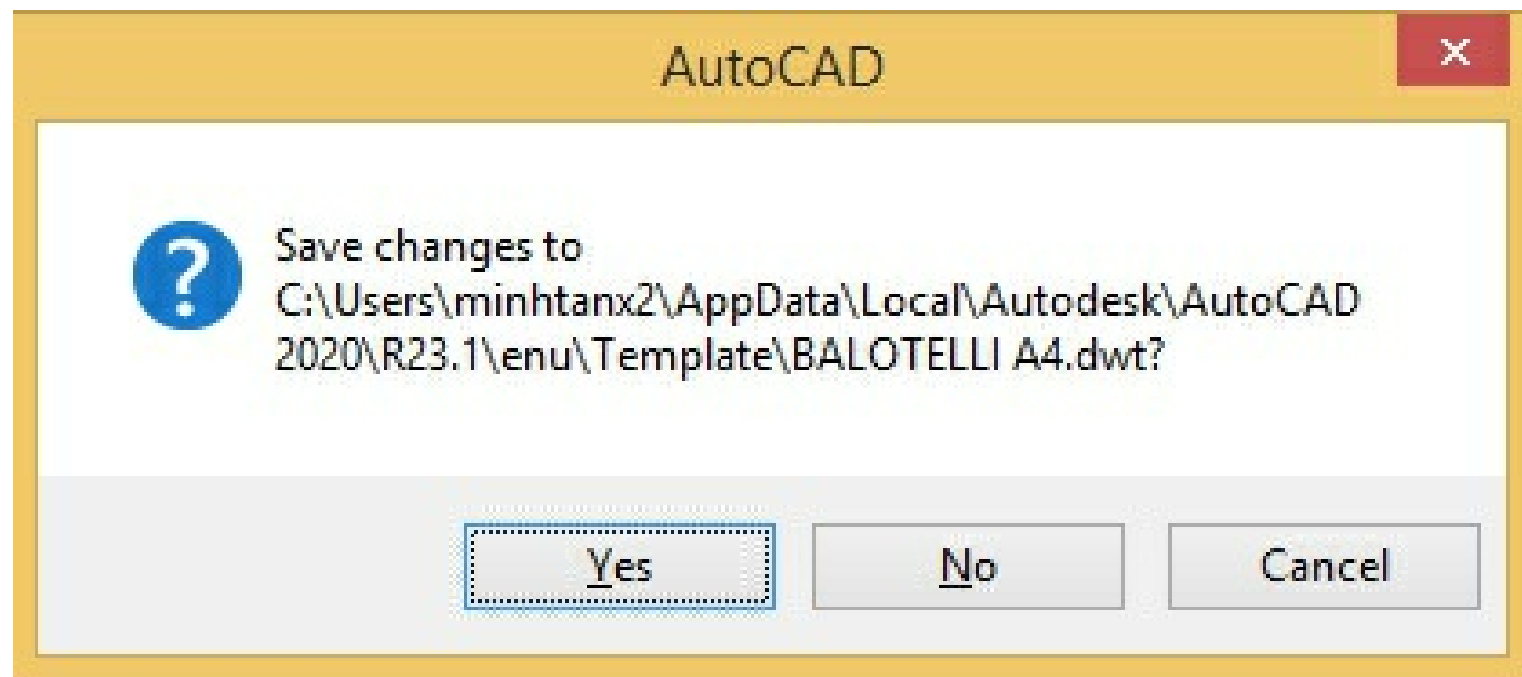
- Start AutoCAD and press Ctrl O
- In the Files of Type section, select the Drawing Template (*.dwt) type.



- Double click on the Template file you want to edit. For example, I want to edit the file “BALOTELLI A4.dwt”.



- Then the file "BALOTELLI A4.dwt" is opened
- You edit the content as you like
- After changing, click Close and select Yes to save.



Thus, the Template file has been modified successfully.

Chapter 19: Workflow With Autocad

I have equipped you with a full range of tools so that you can work with AutoCAD most comfortably. The problem now is to arrange and assemble them so smoothly to have a standard from beginning to end.

Drawing is also a rather time-consuming stage in the project process. But with the method that I am about to reveal to you below, you make drawings for projects that will look like a supercar whirring on the road.

From this moment on, you could forever change the way you work with AutoCAD, be more productive, save time, and do more of what you want. Soon you'll be a giant!

Don't just listen to what I say!

Experience it for yourself to verify!

We are talking about the work of a mechanical design engineer, building product models on 3D software, and exporting projections to AutoCAD. Then proceed to make detailed drawings, present 2D on AutoCAD.

There is an iterative process.

Are you ready ...?

Let's go on stage!

1. Interface settings and optimal settings

The settings include Classic interface, display Menu bar, 3 main toolbars (Layers, Properties, Styles), Options settings, Draft settings, Units.

Set the variable ZOOMFACTOR = 100 to bring the zoom speed, pan the drawing to the fastest (default AutoCAD only equals 60).

Change and add shortcuts command to AutoCAD in a way that suits your style, but still try as much as possible that the default shortcuts command can still be used by others.

2. Build the Template file.

File Template you have made yourself in the above section already.

Create it your way, according to your specialty, whatever you need, just include Layer, Block, Nameframe (1-1 scale), ... Just put everything you need your gear in your tool bag. Don't miss a thing, because there will come a time when you will need it.

3. Export 2D projection data from 3D software (or design 2D yourself right on AutoCAD)

This step you need to do on 3D drawing software (Solidworks, Creo, NX, ...)

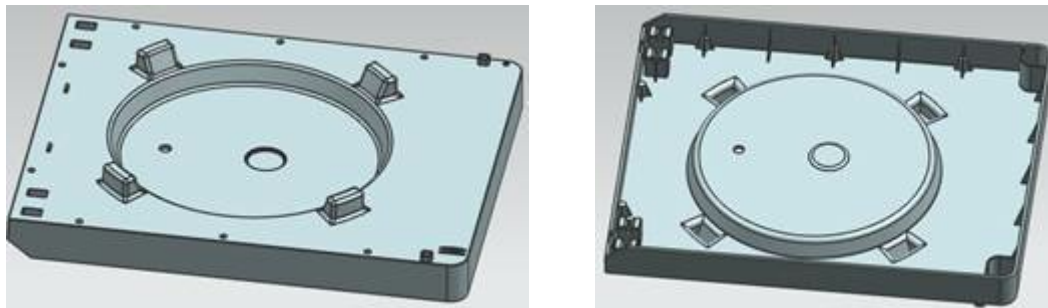
The output of this step is the output projection that meets the requirements:

- The file is exported in the .dwg. file format
- The level of detail is as good as possible, ensuring that all the dimensions of the object are covered
- The projection has a real draw scale of 1:1
- Determine the drawing scale after output (according to the intended paper size)

Note: In case you do your 2D design on AutoCAD, it doesn't matter, your projections must still have a 1:1 real draw scale.

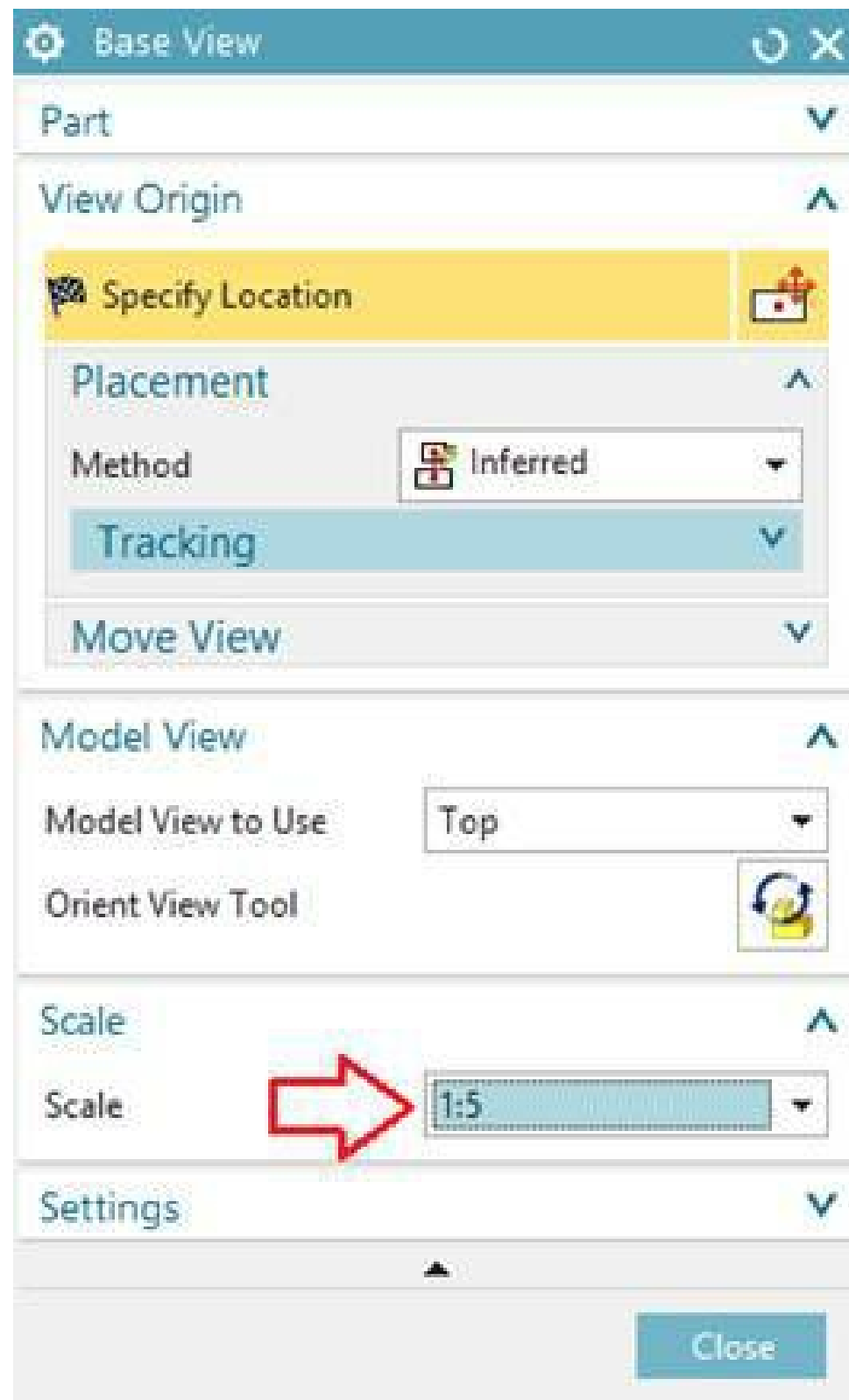
This step of exporting this drawing in 3D software is a stepping stone, but it is very important because it is our input data source. So you need to be careful at this step.

Example: I have a molded plastic part as shown below:

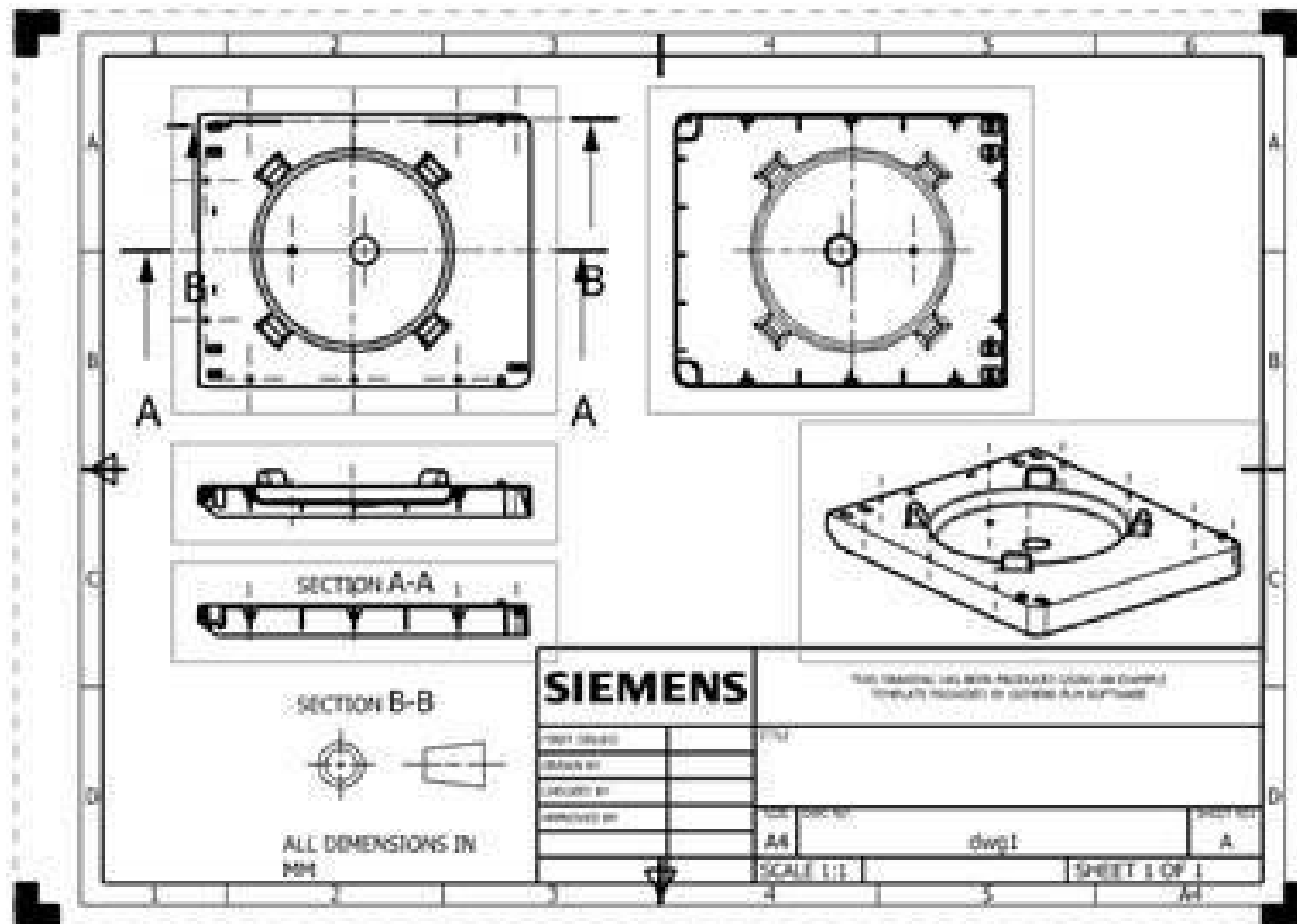


On Siemens NX software, we will output the 2D view as follows (please allow me to say only briefly because it is beyond the scope of the book)

- Open a new Drafting file
- Insert this part file into the newly created Drafting file
- Pull out the views, sections
 - Choose a drawing scale that suits your paper size

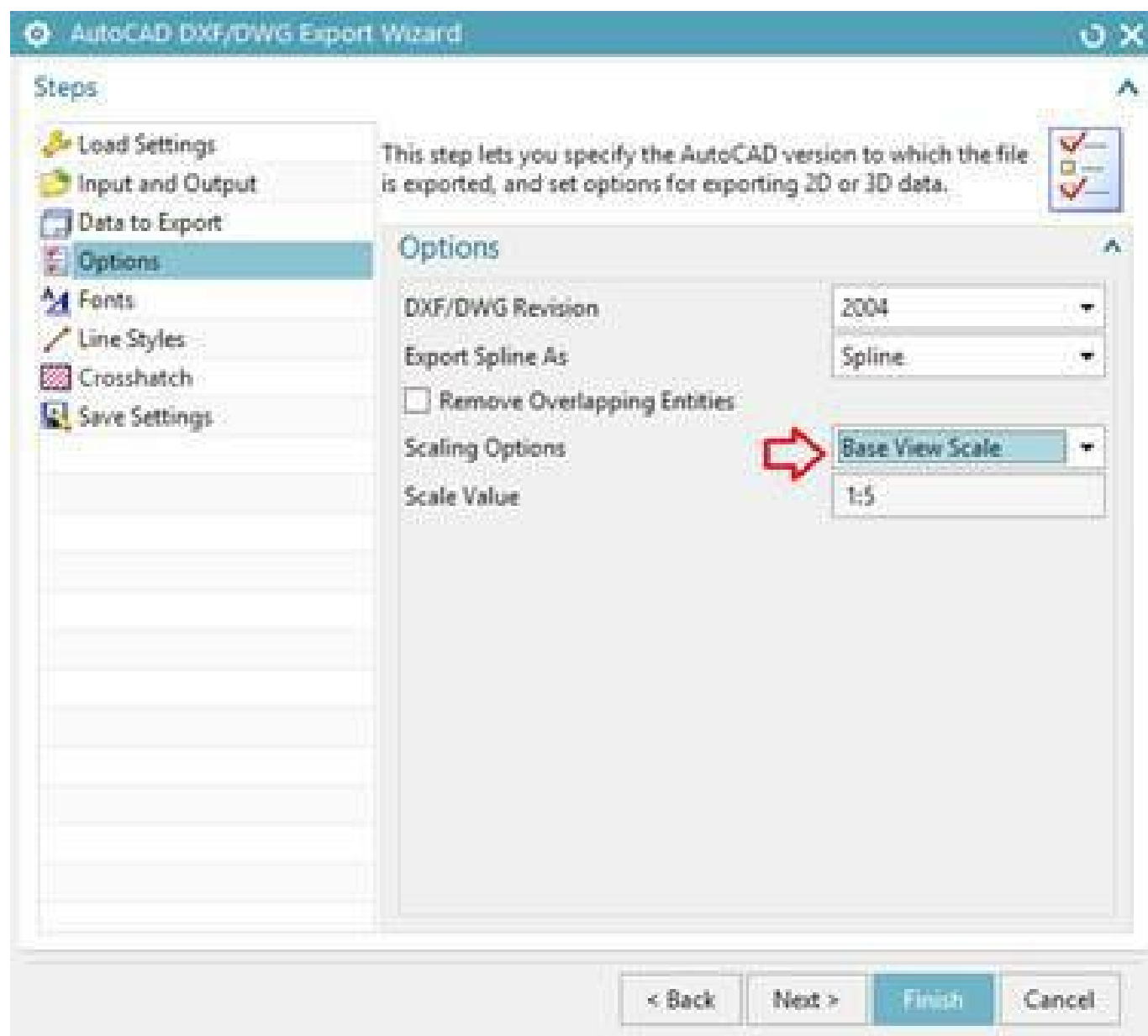


- Try to figure out what dimensions you need to annotate, then output the right views. Because the size is not enough, you may need to re-export the 2D view at least 1 more time. Therefore, in this 2D export step, you must try to do it carefully, with enough projections and sections.
- You also pay attention to the projection method on AutoCAD drawings and Drafting, so the same projection method will be easier to do
- The detail area you do not need to drag out on NX, can process the detail area in AutoCAD very quickly. But it's already lost work on NX, so it's okay to just pull the detail area.
- Don't forget to include the Axonometric View

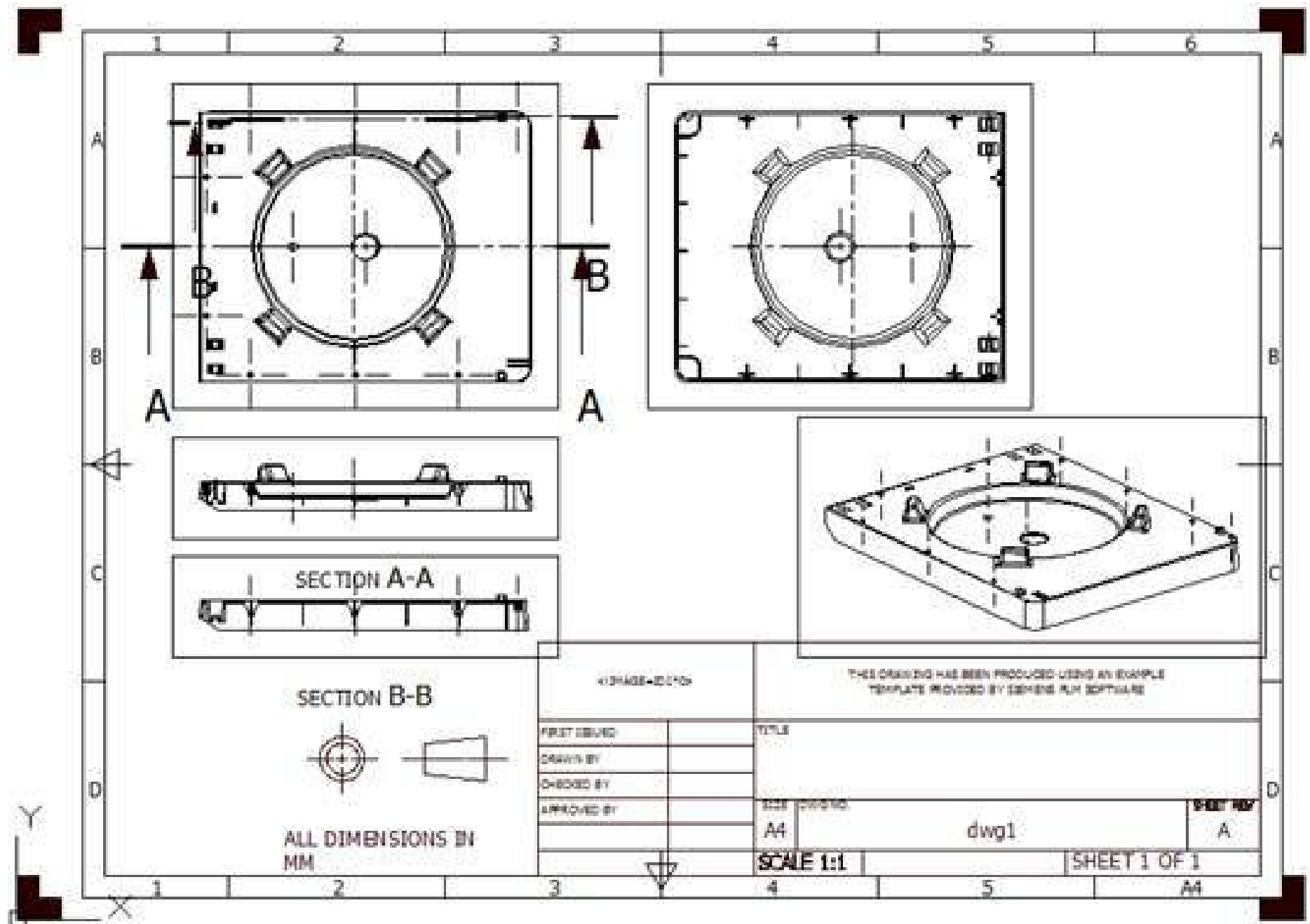


- Export that file to .dwg format

Note to select the Base View Scale option in this section so that the drawing outputs the correct 1:1 **real draw scale**

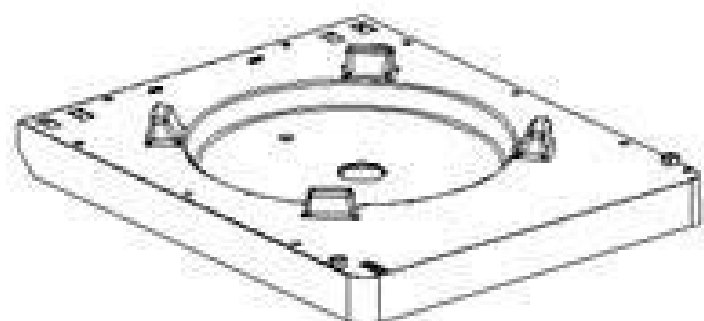
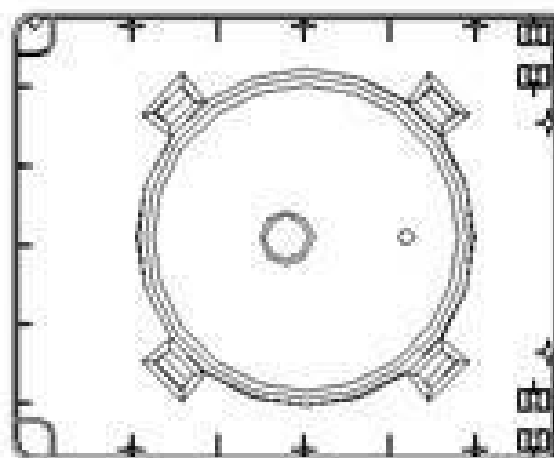
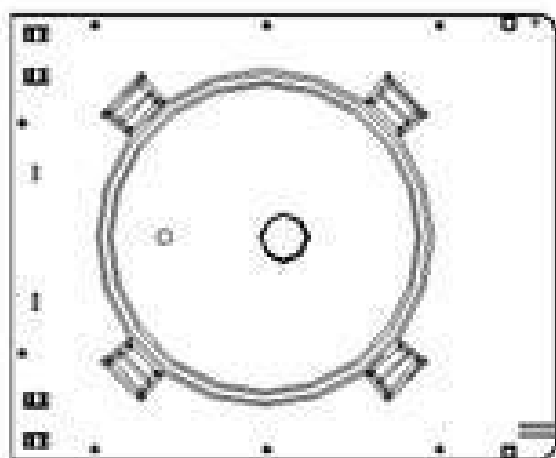


Then we will have a drawing file containing preliminary 2D projections on AutoCAD.



4. Preliminary 2D data processing on the exported .dwg file

- Remove redundant objects (NX name frames, unnecessary whiskers). Can be combined with filter tools (Filter, Selectsimilar) to select and delete faster.
- Move all objects in the view to "layer 0" and set the By Layer option for them.

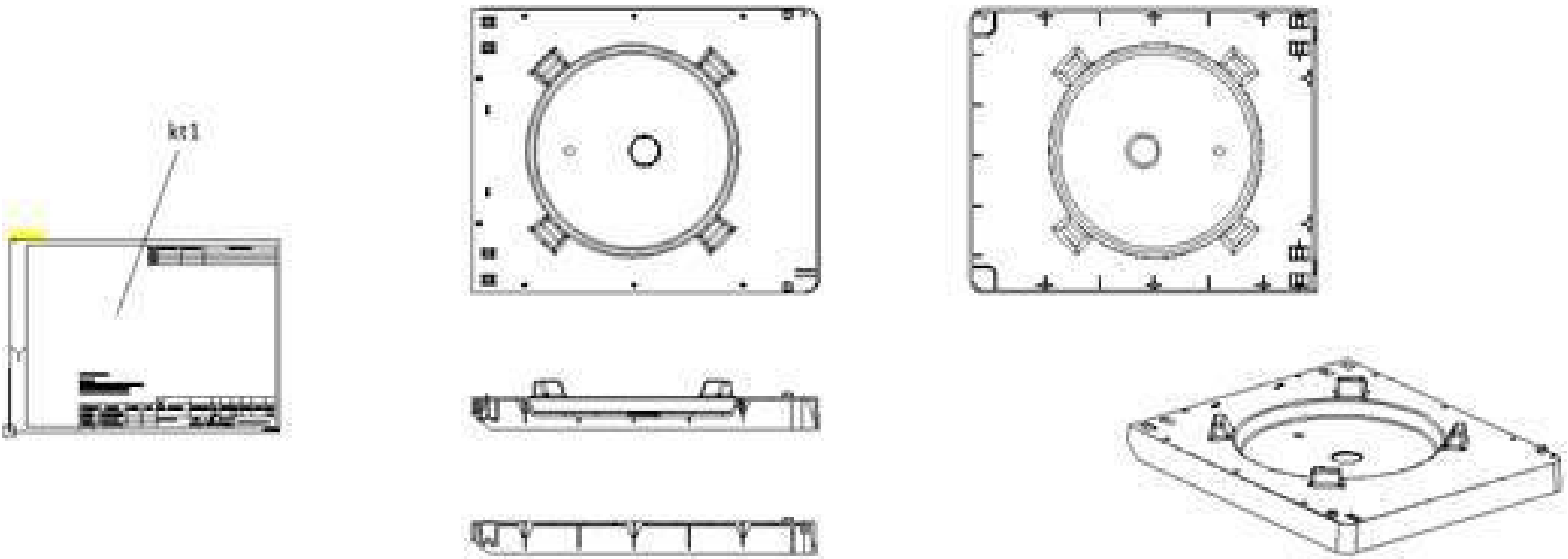


Press Ctrl S to save for sure, assuming the file is called A

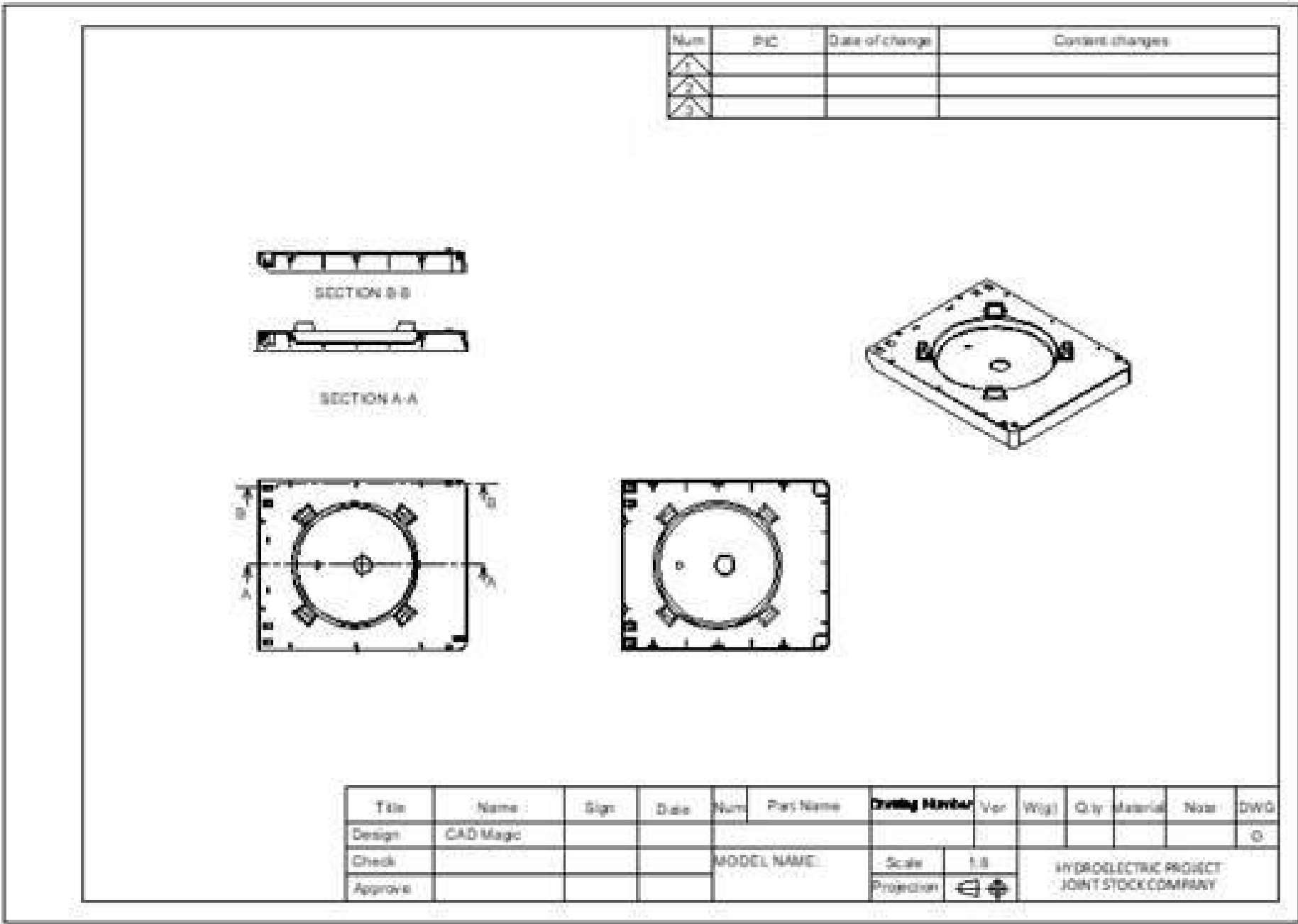
5. Deploy the first drawing in AutoCAD's Model space

We will develop a detail drawing of this plastic part.

- Press Ctrl N to open a new AutoCAD file (must be a new .dwg file created from your Template file), call it the Total file, because later it will contain the entire list of drawings here.
- The title block of the 1:1 drawing scale is already available in this Total file
- Copy data from file A and Paste to Total file.



- In the Total file: keep the original 1:1 drawing scale title block (called "kt1") and Copy it to another named frame (called "kt2").
 - Scale "kt2" according to the drawing scale of the part that has just been drawn.
- For example: Assuming the details in file A have a drawing scale of 1:5 (A4), then we scale the frame named "kt2" up 5 times.
- Move the "kt2" frame to the pasted projection area.

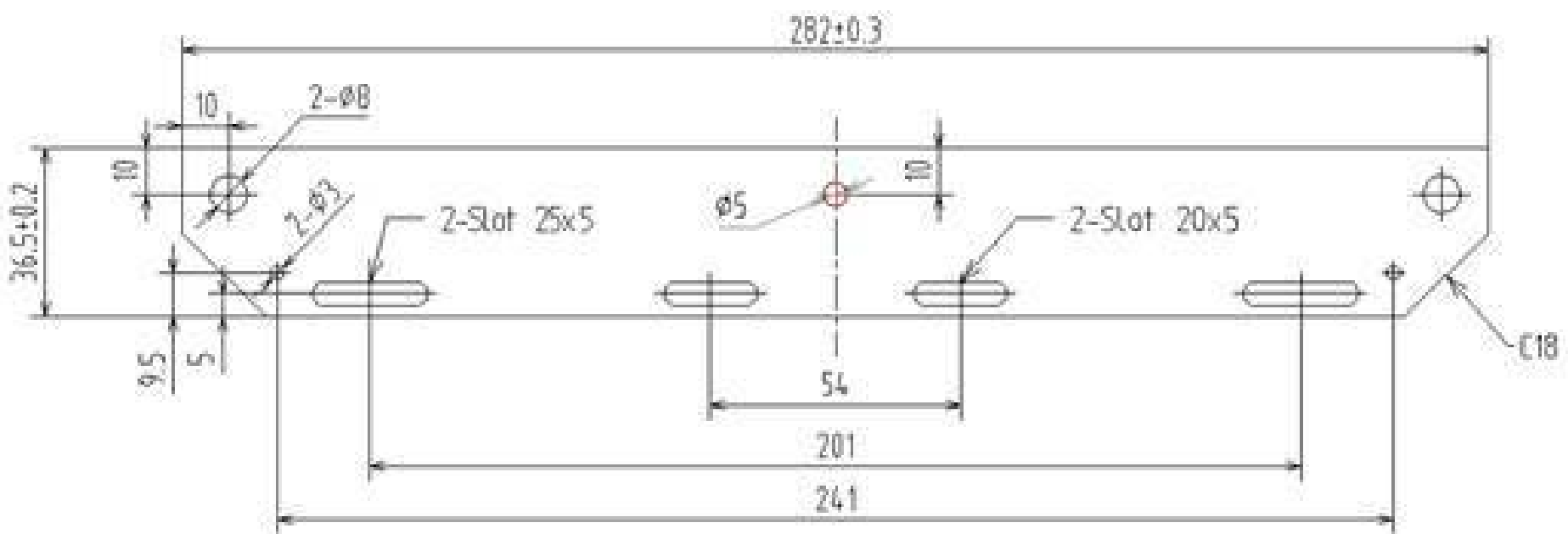


Presenting the content of the drawing

- Arrange the positions of the projections on the drawing so that they are balanced and reasonable
- Write full content on the name frame (detailed name, detail code, material, ...): layer FRAME
- Use the CENTERLINE layer to draw the centerline for the hole, the centerline of the symmetry of the object, the cut line, etc.
- Set the scaling factor for the dashed line based on the drawing scale
- Hatch or retouch hatch: layer hatch
- Note size, tolerance (if necessary) for views, sections: layer DIM
- Note the name of the cut line, section, insert the annotation path (if any): layer TEXT
- Show detail area, size, and name of detail area
- Write Technical Requirements: layer TEXT
- Note the change history (if any): layer FRAME
- Other contents if necessary (list, ordinal number, ...)
- Use the PRINT layer to draw a rectangular frame around the drawing

Some notes when presenting drawings in this step:

Note 1: The positions with the same structure should write the number at the beginning.



Example: 2- $\phi 3$, 2- $\phi 8$, 2-Slot 20x5

Note 2: How to make a beautiful note of the diameter and radius

Sometimes due to insufficient space, we need to record the horizontal hole size like 2- $\phi 8$, $\phi 5$ in the picture above. And it also seems to be more beautiful, directing the eyes of the reader.

How do you want to write that horizontal style?

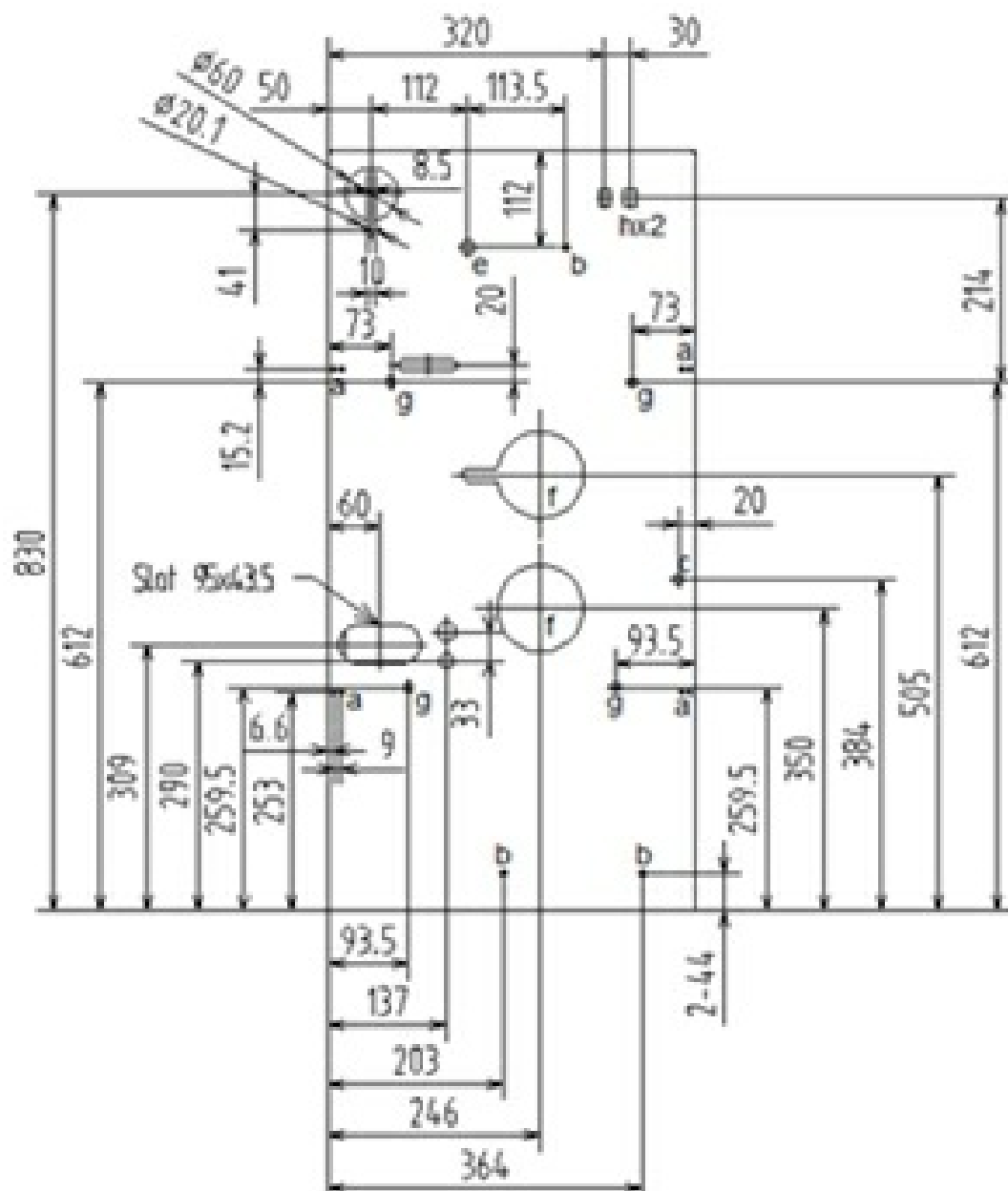
- Before dimensioning for diameter/radius:

- You type the command: **DIMTOH**✓
- At the prompt Enter new value for DIMTOH: type ON ✓

- Then proceed to record the diameter/radius dimension normally. An additional horizontal support line will appear.

- When you want to return to the old default: type the command DIMTOH✓, type OFF ✓

Note 3: With a drawing with many hole systems, each hole system has many holes, while the space for presentation is limited, you can make a statistics table of the hole system.



Hole	Size of Hole (mm)	Q.ty
a	Ø3.5	8
b	Ø6.2±0.1	3
c	Ø12	1
e	Ø15	1
f	Ø100	2
g	Slot 12.5x6.5	4
h	19.2x12.8	2

6. Deploy all drawings in Model space and arrange them properly.

Each machined part will correspond to a drawing, you repeat step 3,4,5 to create a total file, containing all the drawings of the details in that project/product. Types of drawings can be diverse: detail drawings, assembly drawings, breakdown drawings, ...

Arrangement: it's up to you to arrange, as long as you find it reasonable. Drawings can be arranged by assemblies or by type of material (plastic detail drawings, metal detail drawings, ..)

In case a drawing has many Revisions, you should arrange the Revisions to form a vertical line from top to bottom. The highest revision is at the bottom, can add additional notes to avoid confusion.

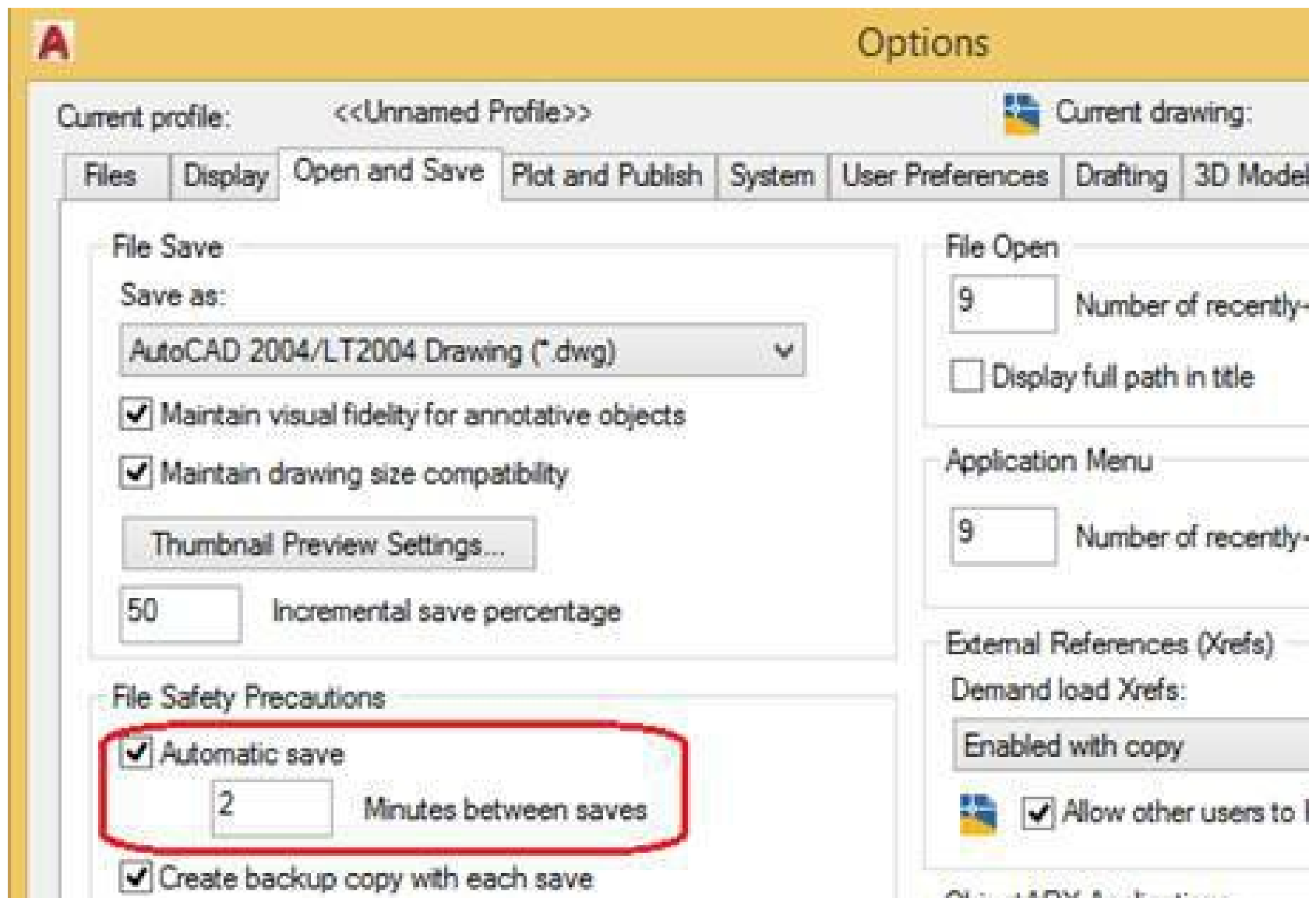
What is the result of this step 6?

It is a list of drawings arranged on the Model in the General file and ready for mass printing.

7. Save the drawing properly

You should actively save after about every 10 minutes of work, it will be better.

You can also set the time for your computer to automatically back up in the Options settings.



8. Mass printing of drawings on Model

Use Lisp “TPL-DNM” to mass printing of drawings on Model

Conclude:

I will summarize the steps in this process to make it more reasonable, because setting up Interface Settings and making Templates you only have to do once, or clicking save drawing is natural.

The process to present 2D drawings on AutoCAD will include 4 main steps as follows:

- Export 2D projection data from 3D software (or design 2D yourself right on AutoCAD)
- Preliminary 2D data processing on the exported .dwg file
- Deploy all drawings in AutoCAD's Model space and arrange them properly
- Batch printing of drawings on Model

Chapter 20: Learn To Be An Expert

Some Tips And Quick Handling Of Common Problems

“It’s important to learn from your mistakes, but it is BETTER to learn from other people’s mistakes, and it is BEST to learn from other people’s successes. It accelerates your own success” - Jim Rohn.

In the Learning Pyramid or Cone of Learning of the American Institute of Education, the most effective method of learning is to teach others.

- You only remember 5% of what is taught through (like teaching and learning in universities today).
- Memorize 10% of what is read from the text (such as reading books, newspapers) and take notes
- 20% through pictures and videos

But can remember up to 90% if you can teach others as soon as you learn.

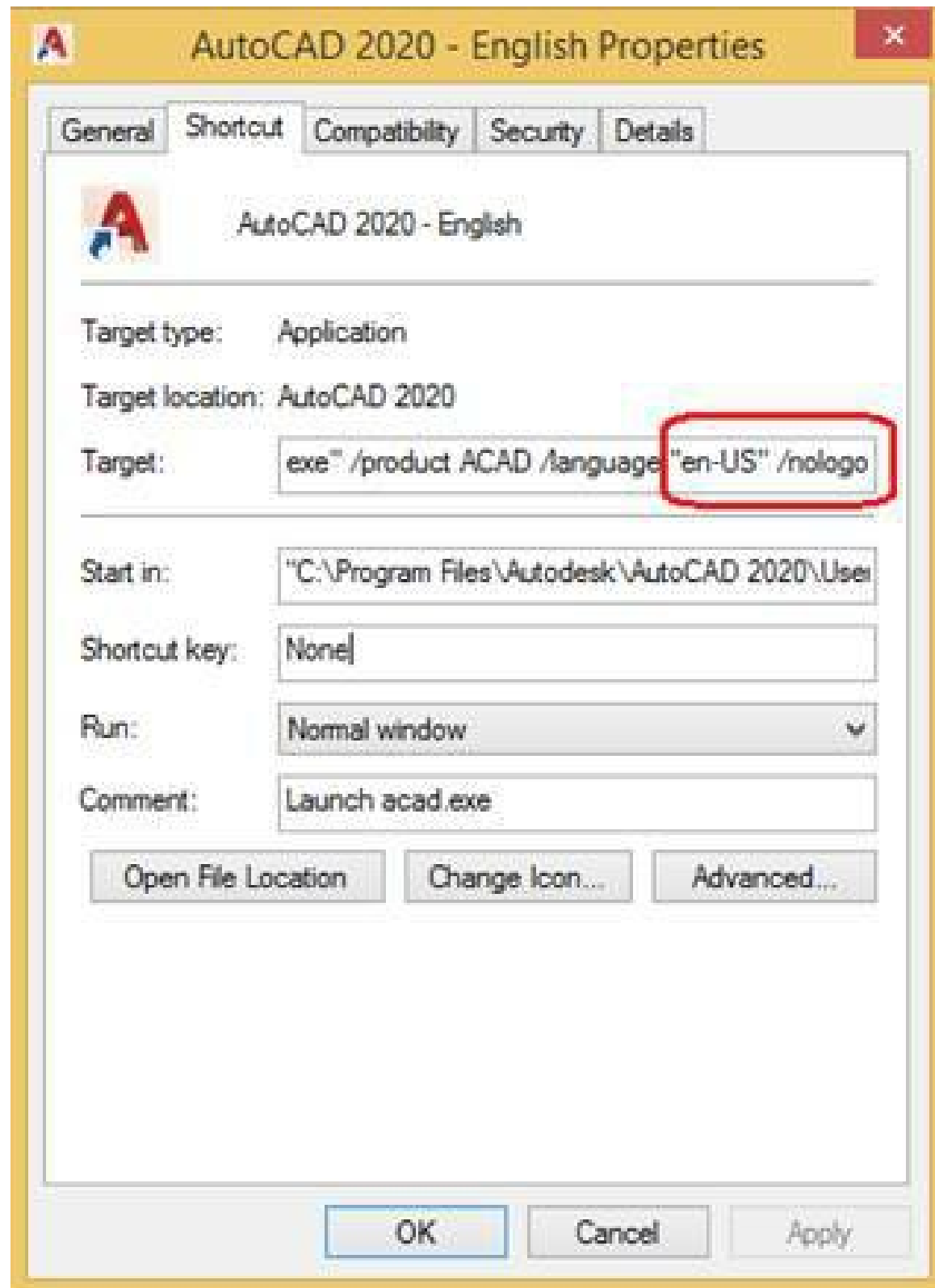
Learning and teaching others, helping them fix problems is the way that has helped me progress with AutoCAD, I avoid making mistakes for myself. Writing this book is also a way for me to remember and consolidate my knowledge.

But don't forget, you only remember 10% of what you read from the text. So with each lesson in this book, I want you to put it right into your current job. Maybe the first time you read it superficially, but read it a second time and practice seriously. If not, there's no way I can help you progress. All my efforts will be like water poured on potato leaves.

Below, I just briefly list a few small tips, some can be called error correction, common or uncommon errors but are quite bony. Errors in AutoCAD are manifold, I can't say, but I will still honestly say "I don't know" if I don't know.

1. How to quickly launch AutoCAD software

- Right-click on the icon of AutoCAD software on the desktop
- Select the Properties tab
- In the Shortcut tab, select the Target line.
- In the Target line, we click at the end of the line, right next to the double quotation mark, write with the following syntax: Space/nologo
- (before "/nologo" is a space)



Click the *Apply* tab and select *OK*

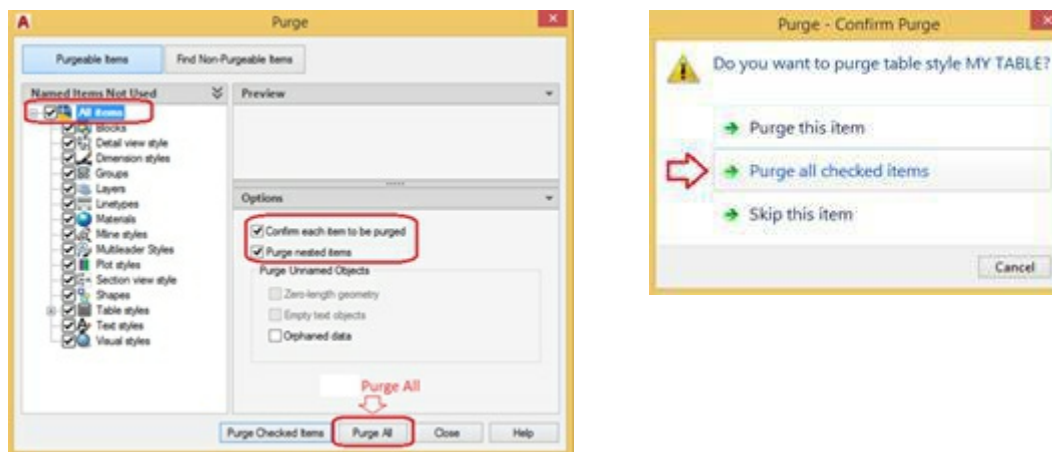
Combined with making your Template file the default file at startup

2. Clean the drawing

A drawing is only considered complete after you have cleaned it, i.e. removed all excess (drawing junk). This will help optimize the drawing capacity and make the drawing smoother. For computers with moderate configuration, reducing the capacity will make a noticeable change in the zoom pan speed.

The way I usually clean drawings is as follows:

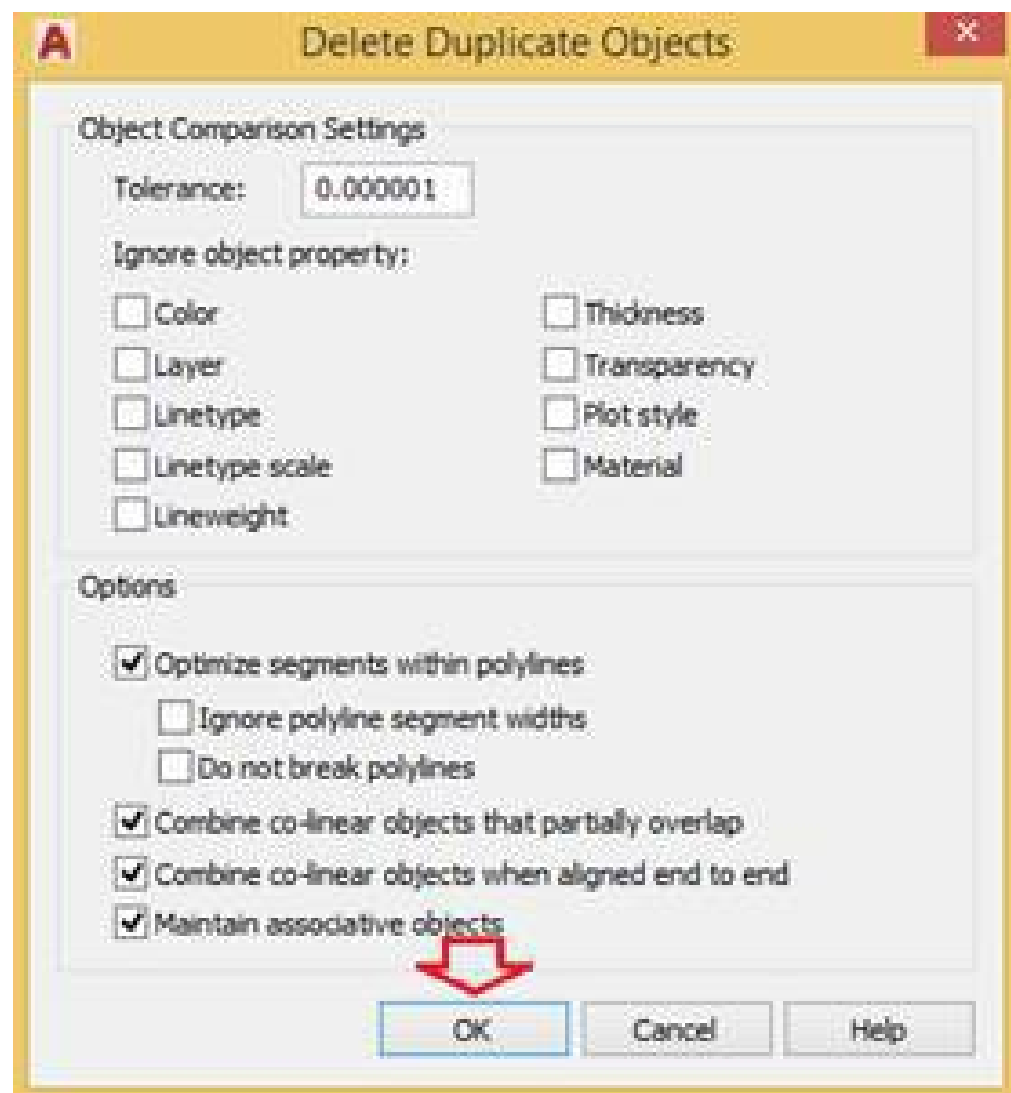
- Type the command: **PU**✓ (Purge) command to preliminarily remove redundant objects. An extra object is an object with a (+) sign at the beginning of a line



- Type the command **OVERKILL** ✓

At the Select Objects prompt: you scan all existing drawings and objects in the Model space, type Enter ✓

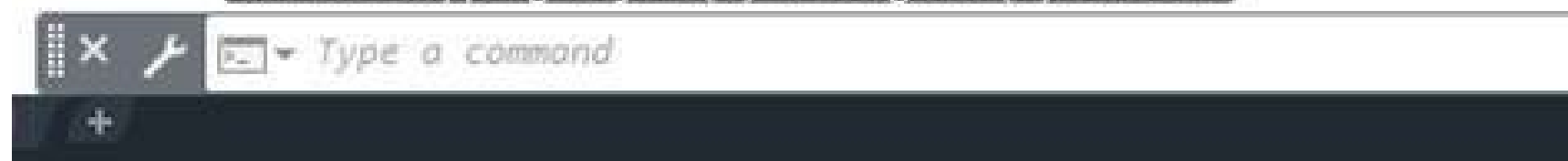
- Click OK to remove duplicate objects



Select objects:

2 duplicate(s) deleted

275 overlapping object(s) or segment(s) deleted



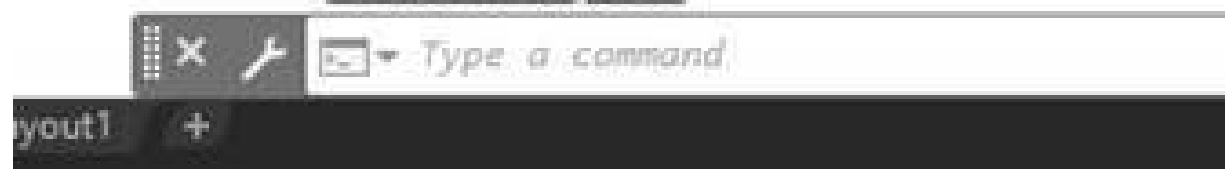
- Type the command AU✓ (AUDIT)

- At the **Fix any errors detected prompt**, select Y✓ (Yes) to correct potential errors on the drawing if any.

Auditing AcDsRecords

Total errors found 0 fixed 0

Erased 0 objects



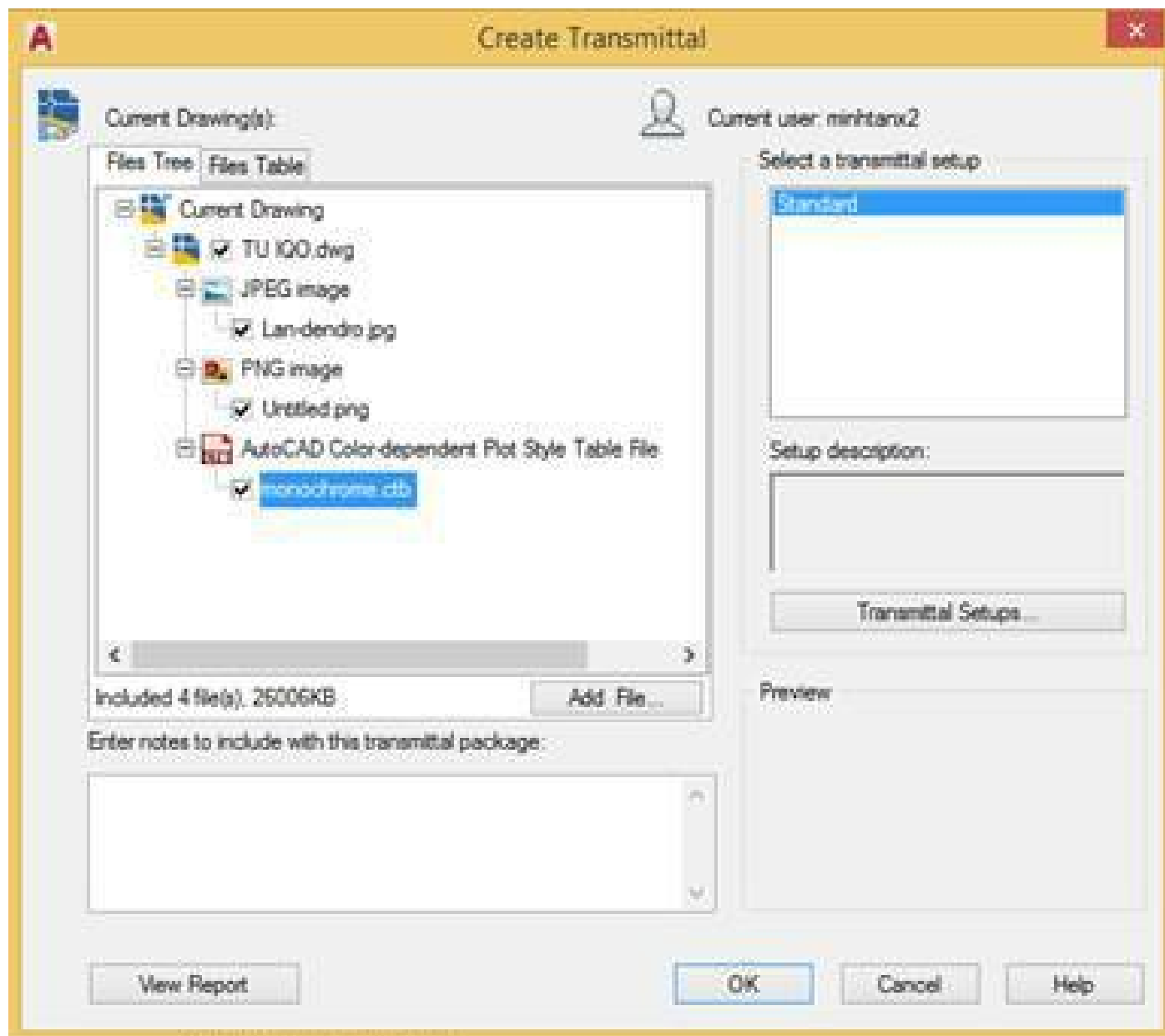
- Type PUC command a second time to be sure to clear all garbage.

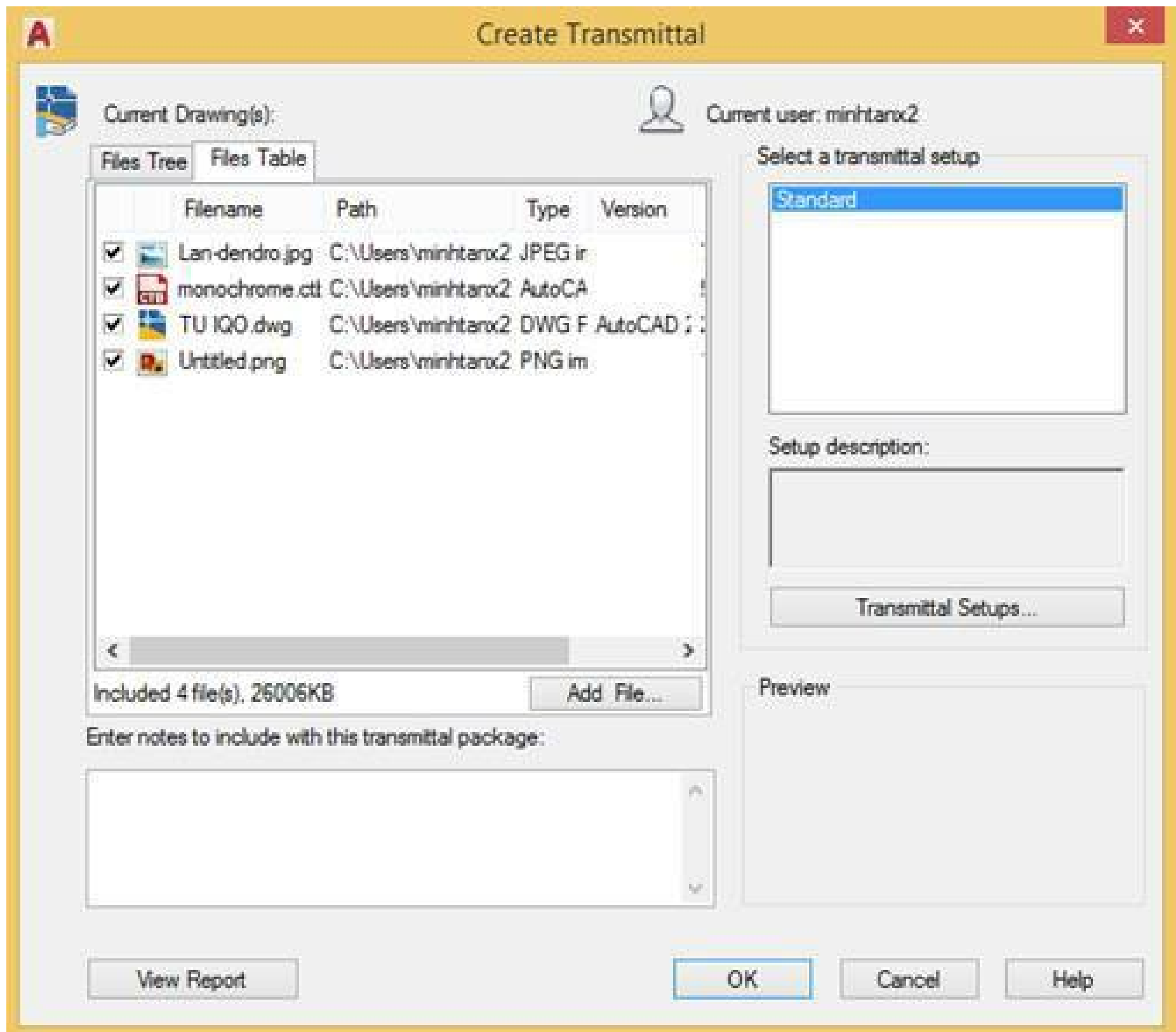
3. How to save drawings safely when there is an image object inserted.

In some cases, the dwg file has an image inserted inside, when sent via email or copied to USB, the image may be lost. Therefore, to be safe when sending drawings with images, we should compress them into a zip file. When you send the drawing, the zip file will be sent.

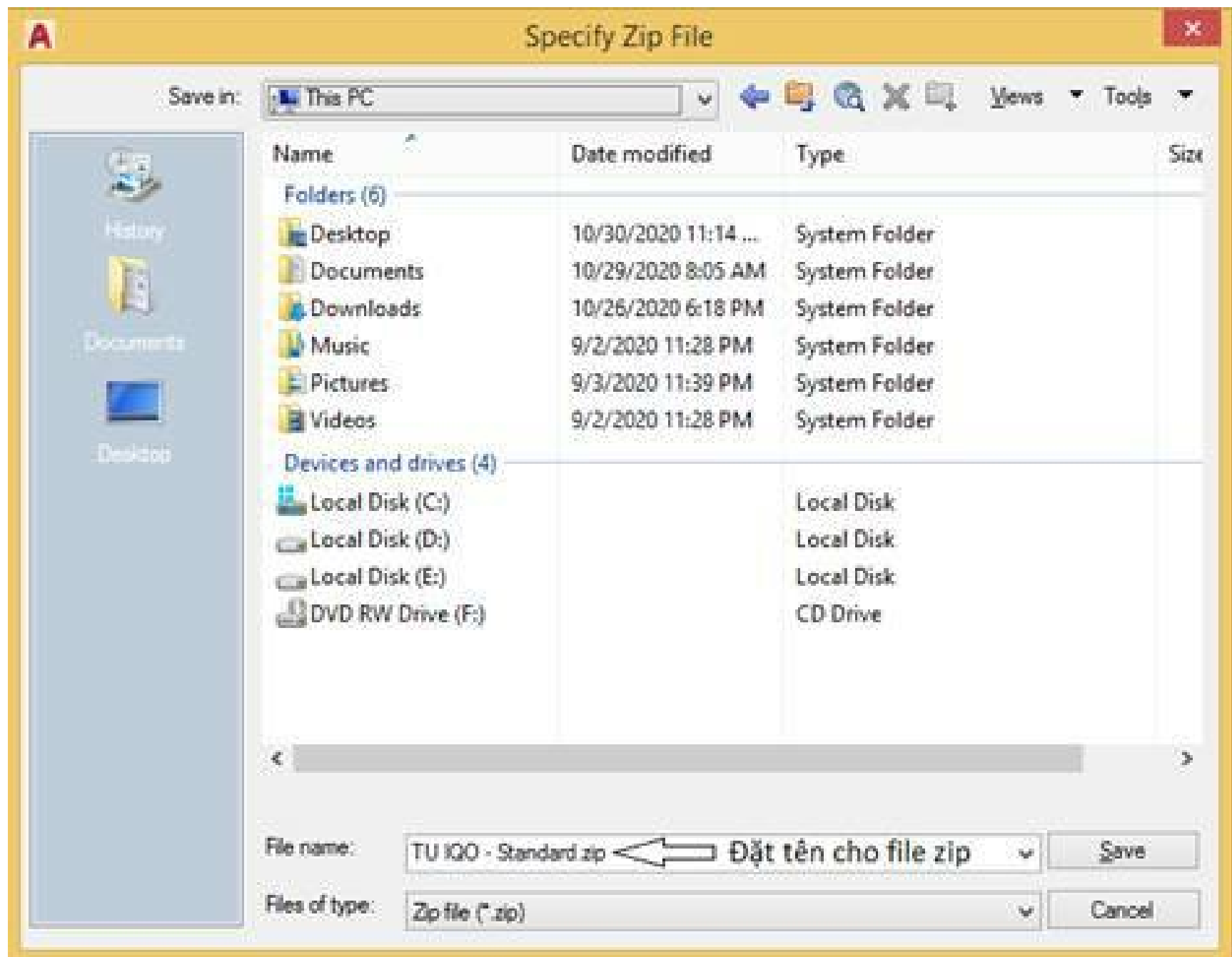
You type the command: **ETRANSMIT**

Images and stroke files are included





- Click OK to name the .zip file and save it.



When you need to send it, please send this zip file.

4. Error of data loss on file

This is a rare but not uncommon error, this is an extremely serious error. Because your data is so important, it's often the result of months of work. If all your efforts are lost, it's too dark and boring. No one wants to be a moron.

Error phenomenon: The test file is still large, but opens with nothing, even though I have done all the commands to display the layer, it does not change the result.

The treatment is as follows:

- Open the file itself or you can open any new file
 - Type the command **RECOVER**✓
 - In the Select File dialog box, find the path to the file itself on the drive / Select Open
 - AutoCAD will ask: "Would you like to open the file read-only?", you choose "Yes"
- => Now that your file is restored data, you will see the restored data on the screen.
- Choose File / Save As to save it with a different name.

5. I want to completely delete a layer but can't delete it, even though I have purged many times and go to the layer management panel to select Delete but still can't delete it?

Answer: You use the LAYDEL command and select any object belonging to that layer to completely delete the layer.

6. Add font for AutoCAD what type?

Text used on AutoCAD drawings is usually used by two types of fonts:

- Truefont: font of the Windows operating system, located in the Fonts folder in Windows.



You look at the Type format to know it is Truefont

Name	Type	Size
 _VnAvant	TrueType font file	38 KB
 _VnAvantH	TrueType font file	34 KB
 VHANTIQ	TrueType font file	62 KB
 VHARABIA	TrueType font file	74 KB
 VHARIABI	TrueType font file	35 KB
 VHARIAL	TrueType font file	42 KB
 VHARIALB	TrueType font file	43 KB

You just need to Copy and Paste your truefont into this folder and it will automatically be installed in Windows. AutoCAD will use the same Truefont with Windows here.

- Font SHX: is a font format exclusively for AutoCAD. With these fonts, just copy them all and paste them into the Fonts folder in the software's installation directory.

How to quickly find the software's installation directory:

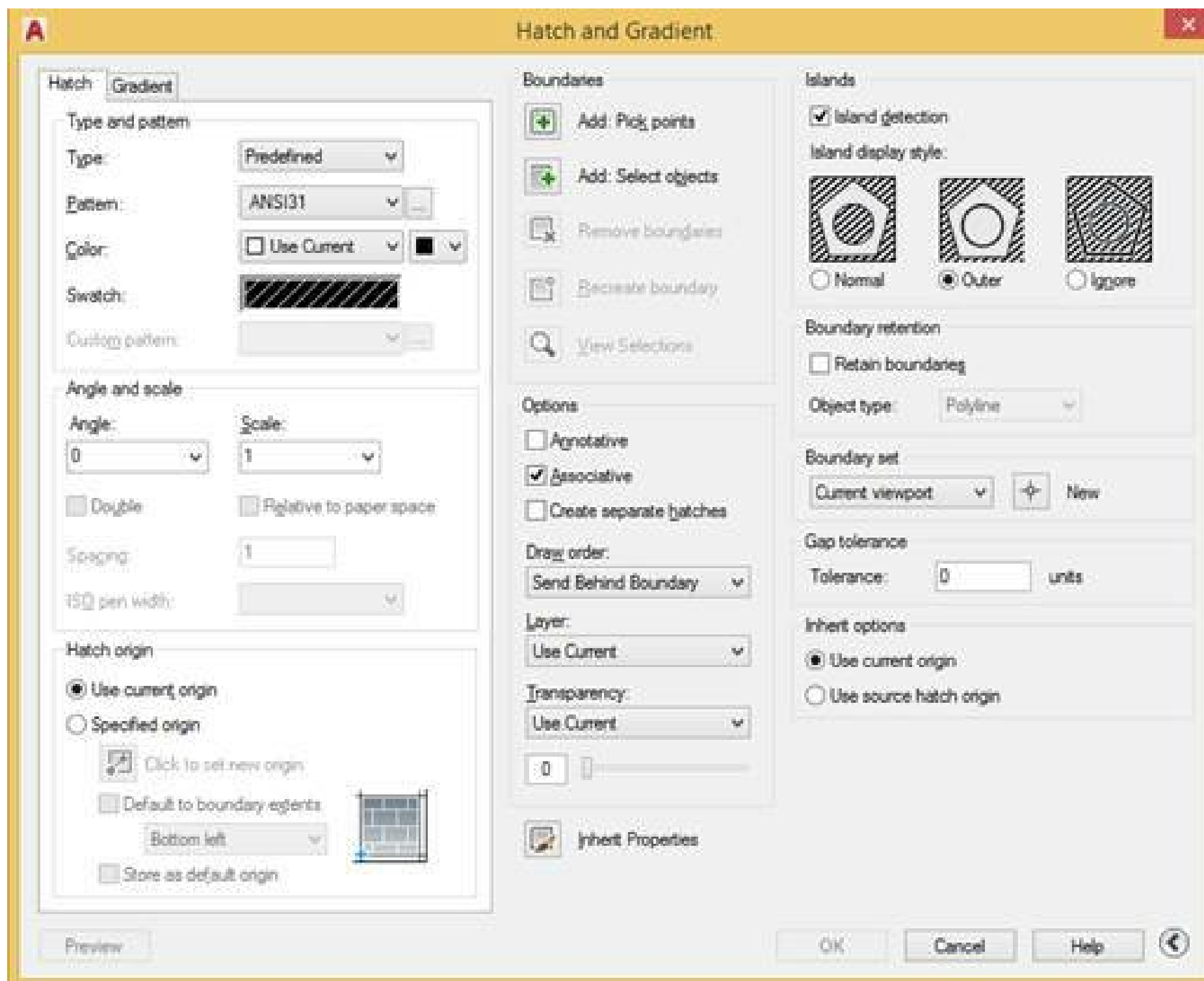
Right-click the AutoCAD icon on the Desktop/ Open file location/ Fonts/ Paste the .shx fonts you need to add to this folder.

This PC > Local Disk (C:) > Program Files > Autodesk > AutoCAD 2020 > Fonts		
Name	Date modified	Type
 @extfont2	8/9/2018 11:03 AM	AutoCAD Compiled Shape
 3CBRUS1	8/9/2018 11:03 AM	AutoCAD Compiled Shape
 3CBRUS2	8/9/2018 11:03 AM	AutoCAD Compiled Shape
 3CCOUR	8/9/2018 11:03 AM	AutoCAD Compiled Shape
 3CELIR	8/9/2018 11:03 AM	AutoCAD Compiled Shape

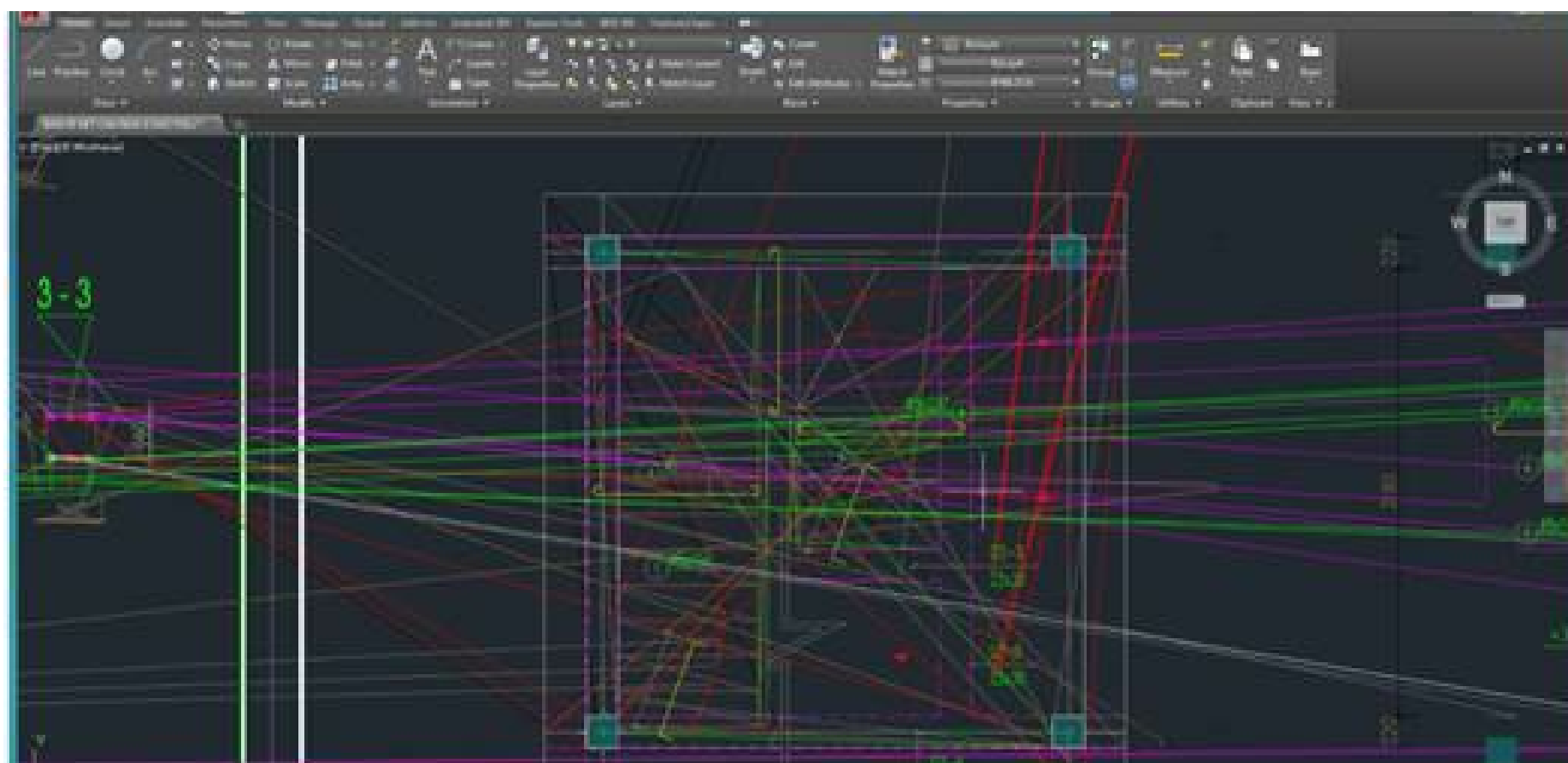
7. I type the H command to perform the Hatch section, but the Hatch and Gradient panel doesn't appear as usual? How to make that table appear?

Answer:

You type the HPDLGMODE command, set the value to 1 or 2, then retype the H command to see the Hatch and Gradient table.



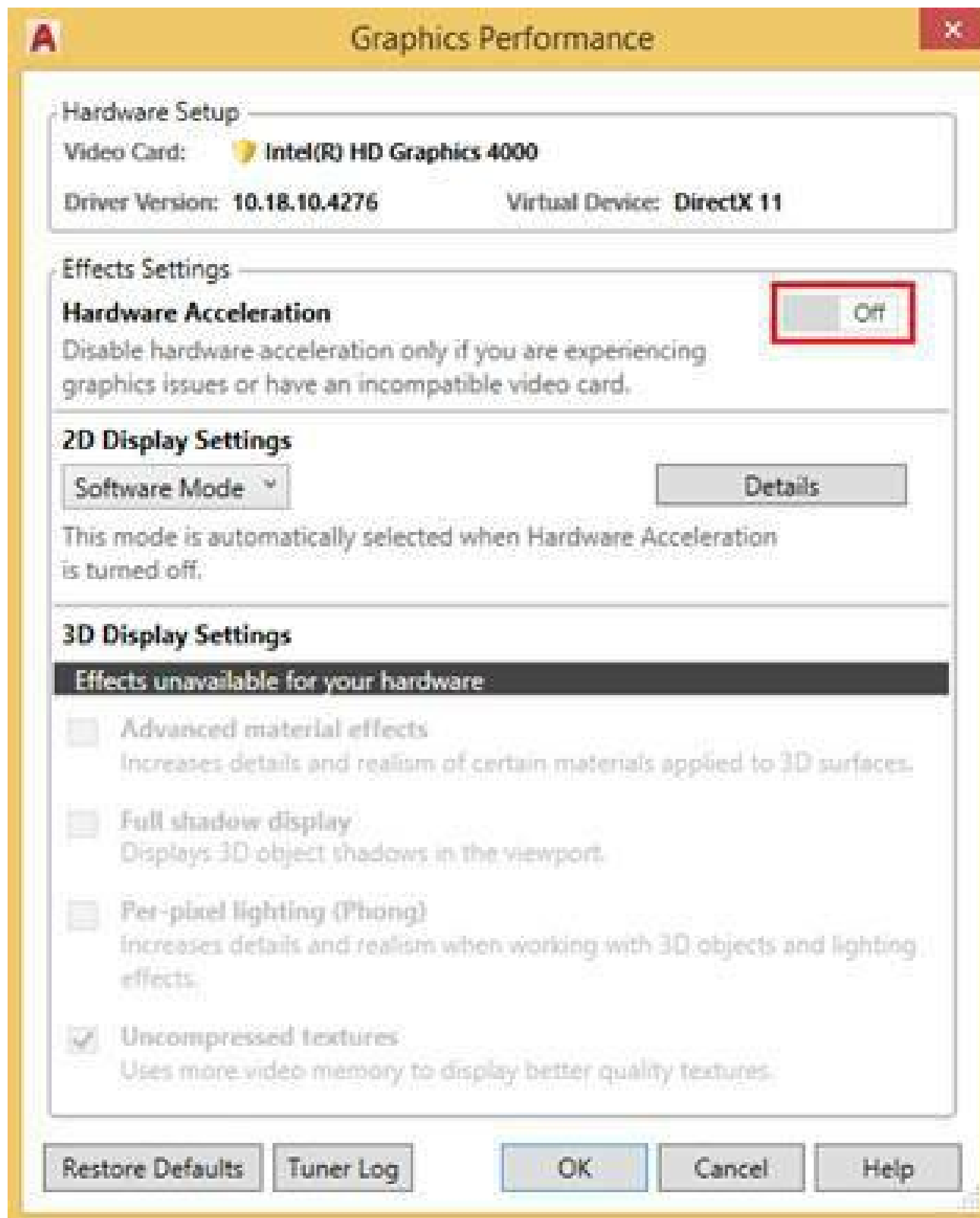
8. After opening the AutoCAD file, the screen displays as tangled as strings. Same file, but I open it on another computer and it works fine. How to fix it?



It is an error due to an incompatible video card.

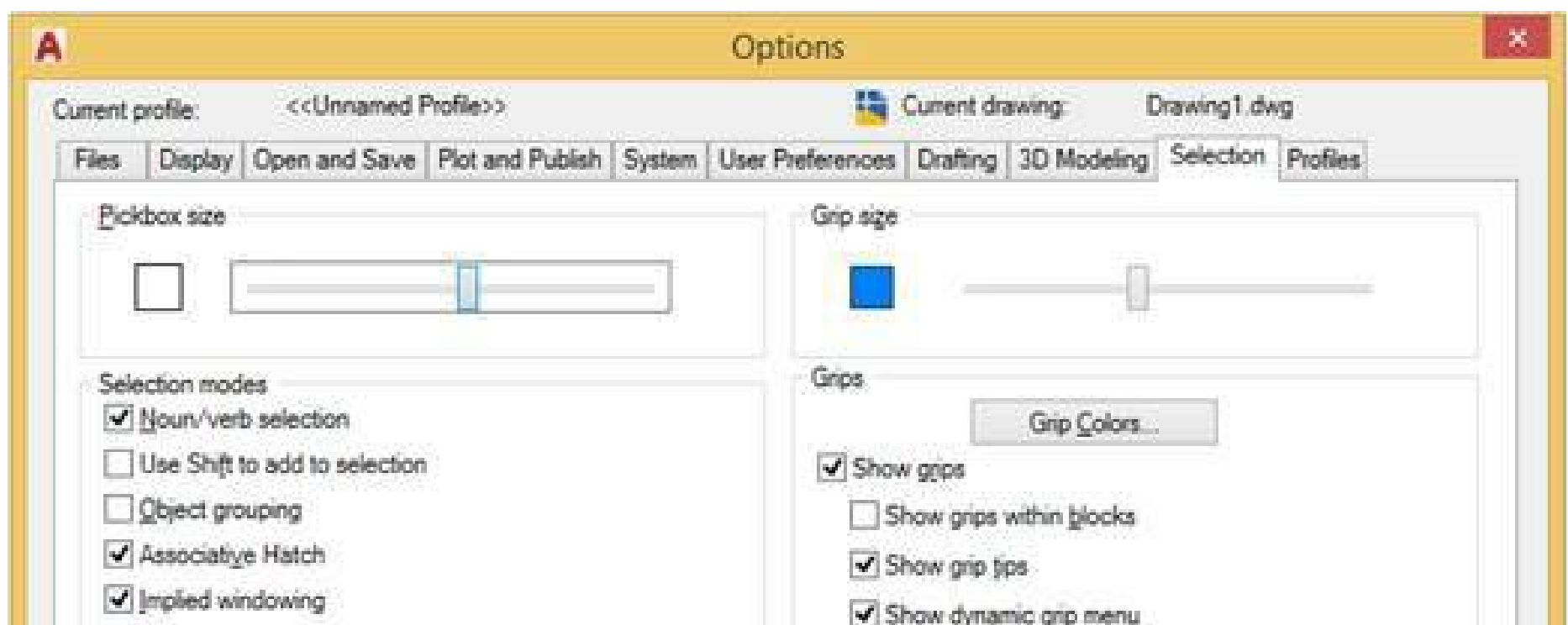
How to fix:

- Go to *Option/ System/ Graphics Performance/* in *Hardware Acceleration* option, you switch from ON to OFF state



9. I want to edit the text content, but when I double-click on the text to edit it, it can't be edited. How do you overcome it?

You go to *Options / Selection /* check the box *Noun / Verb selection* to edit



10. Why when I use the DLI command to record the dimensions, it takes an extra step of pressing Enter to agree. At the same time, the dimension line after appearing on the screen is decomposed into discrete objects.

Reason: you accidentally set the system variable **DIMASSOC** = 0

How to fix:

Type the command DIMASSOC ↵ and enter a value of 1 or 2.

It is better to enter 2 so that the dimension line is always associated with the object.

11. Normally, I just double-click on the Block to enter the Block Editor environment to edit. But there are some Blocks that I also double-click but can't edit and it asks me to enter the content again. How do I fix these Blocks?

Blocks have many types. The block you are describing has been assigned an Attribute attribute, so you cannot double-click to edit it like normal Blocks. However, no matter what type of Block, you can edit it with the **BEDIT** command.

Conclusion and Thank you

Today, the presentation and printing of AutoCAD drawings is an important part of every engineer's requirement. Asking us to know how to make the most hard work and the least time. With many years of design experience on AutoCAD, I wrote this book with all the heart and love. I really hope you will apply my heartfelt sharing to your current work. And what I want more than you not only apply it skillfully but also know how to use it in a flexible and creative way in many different cases of work problems, that means you have mastered completely.

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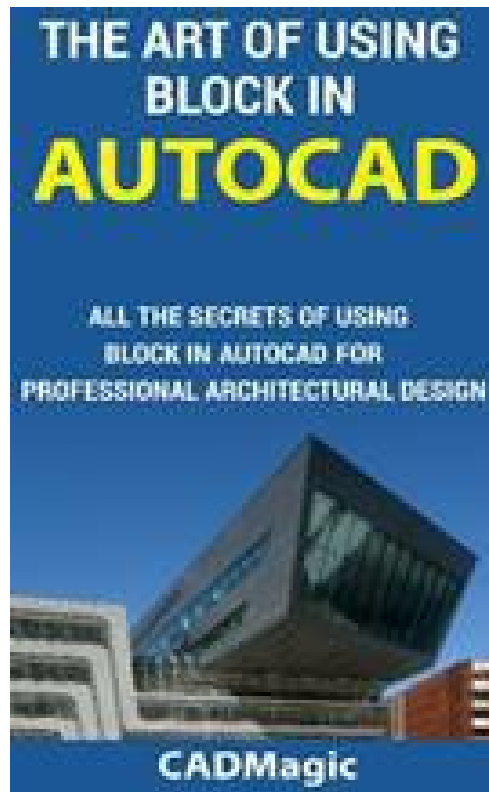
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Thank You and Best of Luck to You!

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If you are a professional engineer, often have to work with AutoCAD and Block drawings. But you have not completely mastered in-depth knowledge about Block. I would recommend this special book, which is my passion, I am really impressed with it. Referring to this book, I think it is one of the best books about Block in AutoCAD for you. If you are interested, please read it, I believe you will be satisfied!

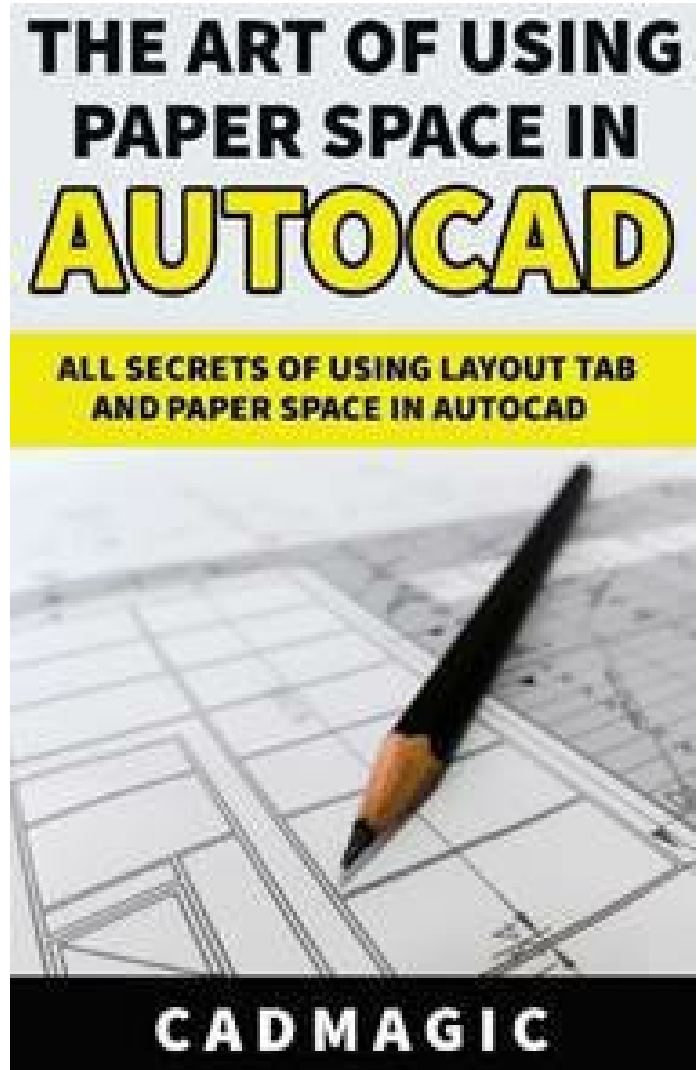


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