AutoCAD Basic Commands

Suitable for Grades 6-12

In This Tutorial You Will Learn…

- How to type commands in AutoCAD
- How to make **new layers**
- How to draw **lines**
- Drafting Settings
- How to **offset**
- How to **erase**
- How to draw **circles**
- How to make **fillets**
- How to **trim**
- How to **hatch**
- How to draw **rectangles**
- How to write **text**

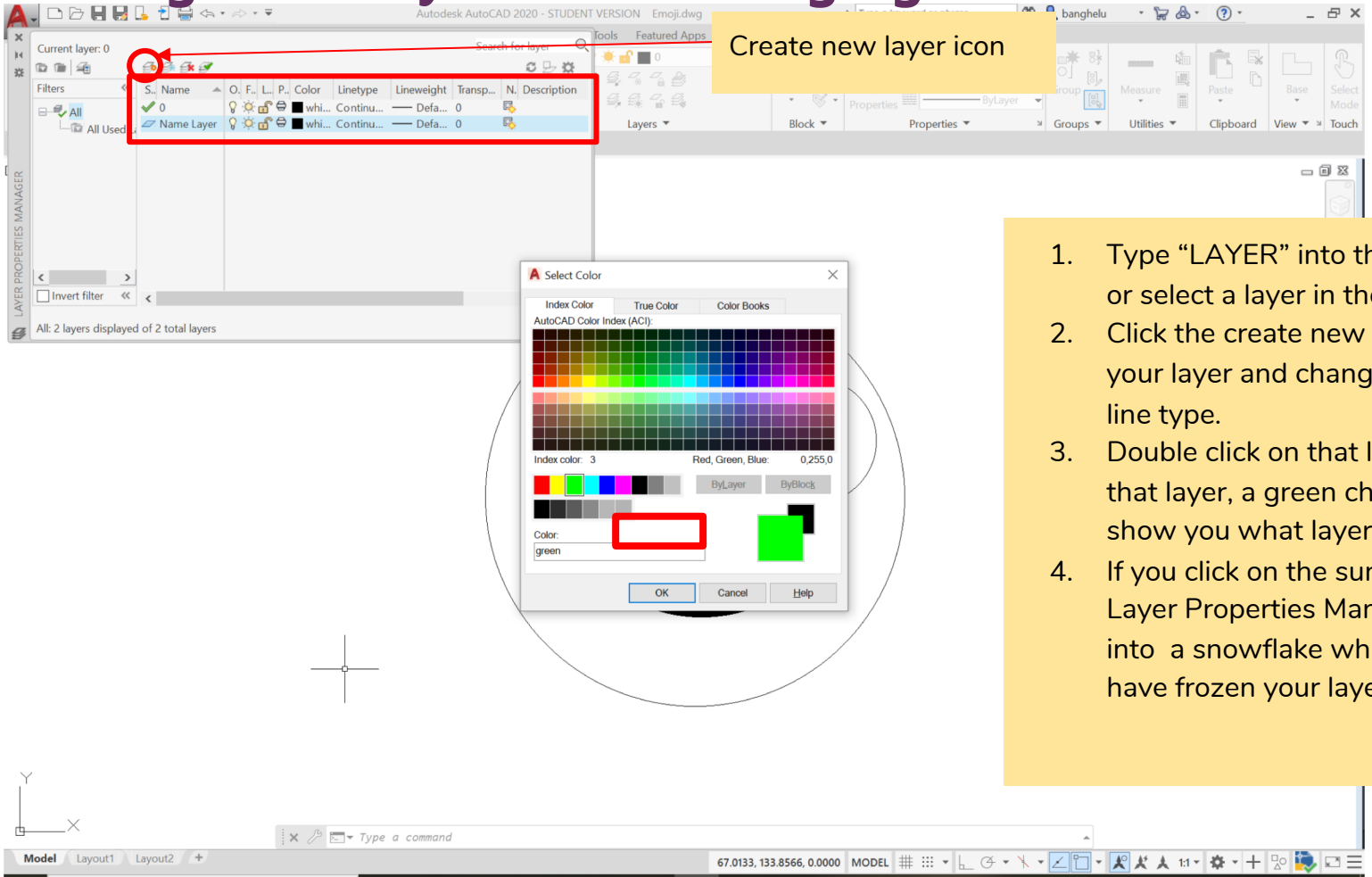
Commands in AutoCAD

- Commands tell AutoCAD what to do and draw
- Commands are typed into the command bar
- In these tutorials, commands are written in all capital letters
- If you are ever unsure of what you are doing or do not want to be using a command you have inputted, press the escape button on your computer (“Esc”)
- Check out the “AutoCAD Commands Cheat-Sheet” for a list of commands that may be helpful for you

LAYER

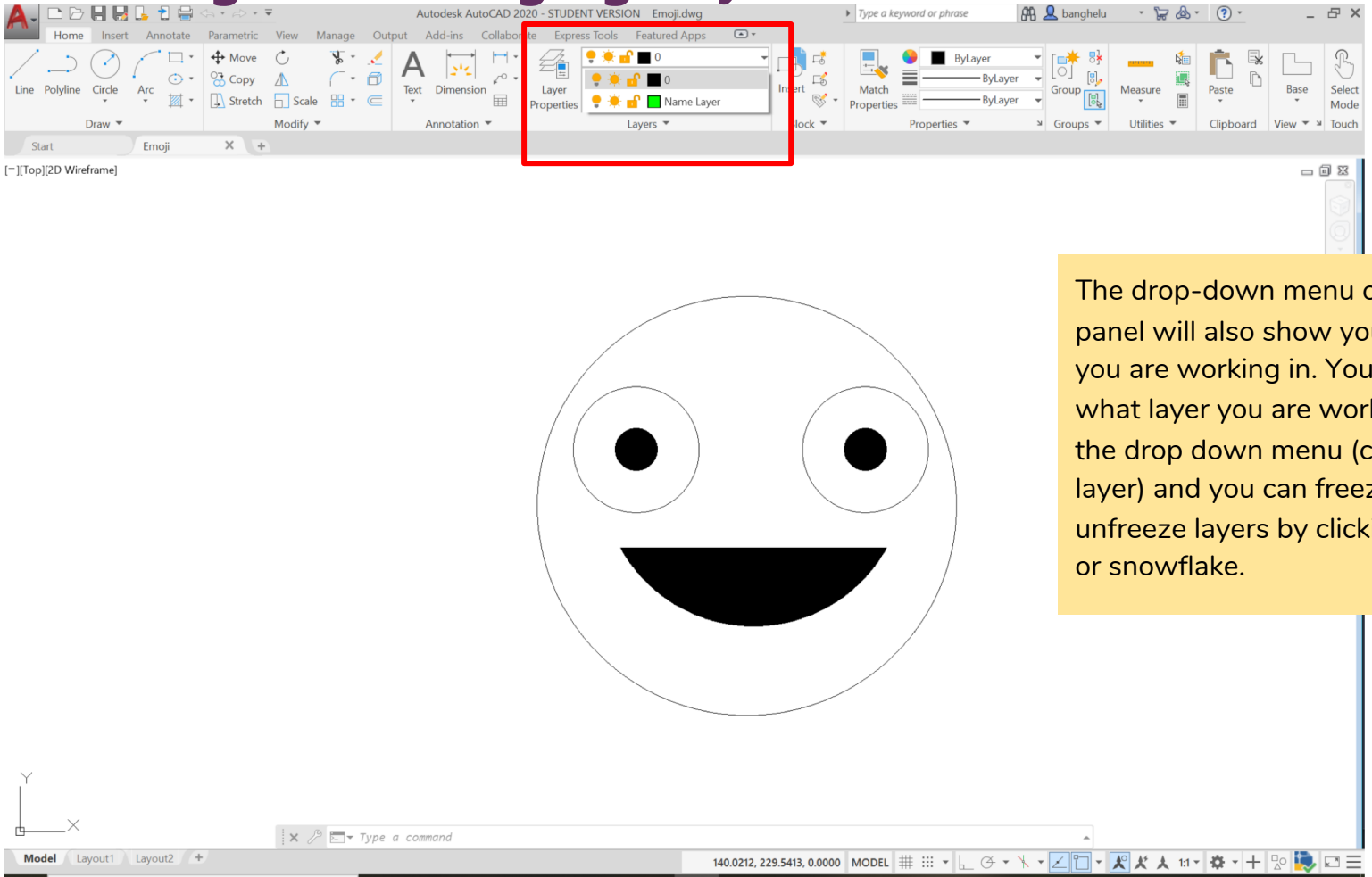
- When creating a drawing, you can work in different layers
- Layers are different parts of your drawing that overlap to create one complete drawing
- By creating different drawing layers in the “Layer Properties Manager” you can change the line colour, thickness, and type in each layer you create
- This way you can have drawings with lines of different colours, thicknesses, and types
- Managing your layers is very important to keeping your drawing organized

Making New Layers and Changing Line Colours



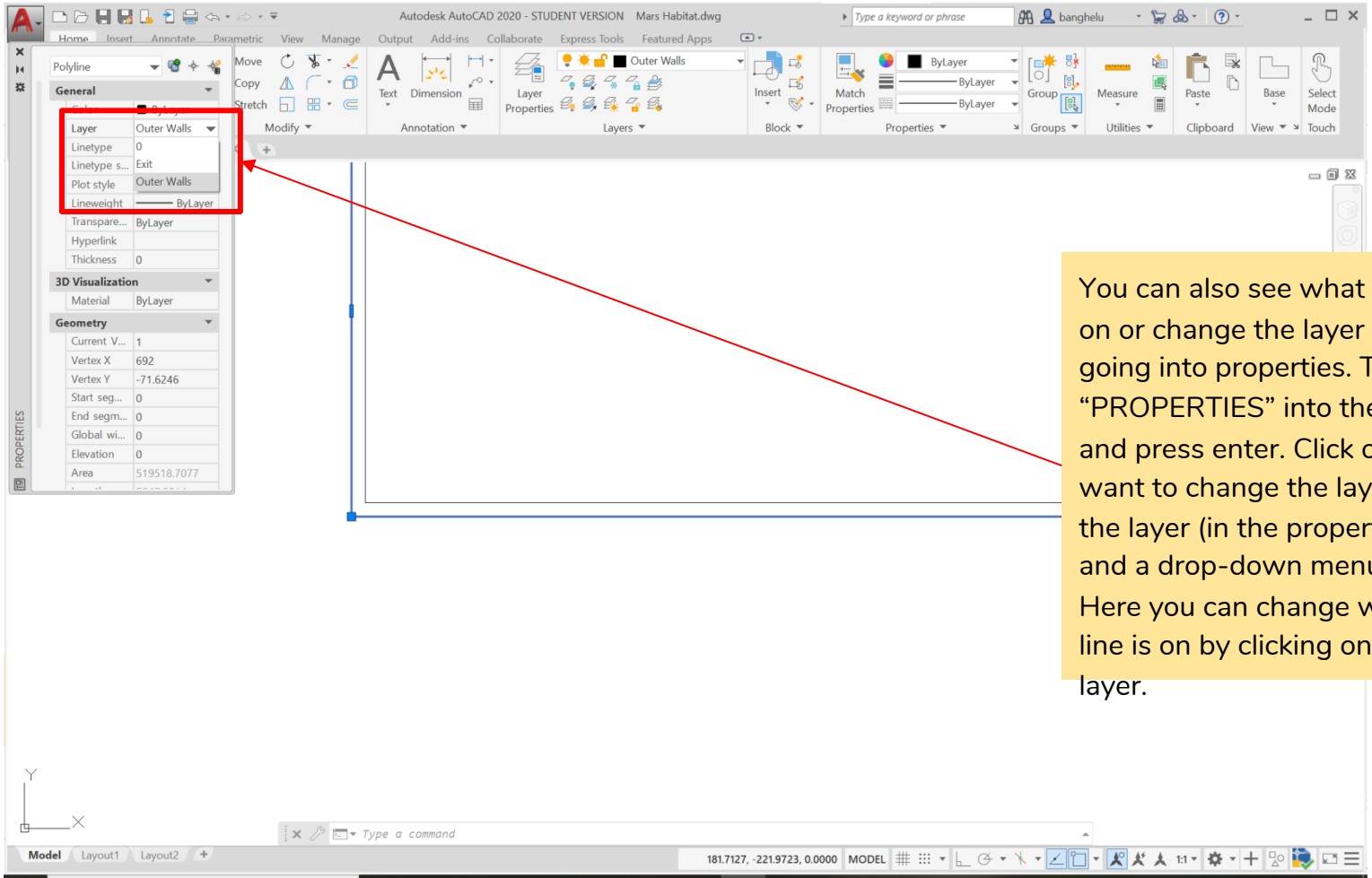
1. Type “LAYER” into the command bar or select a layer in the layer panel.
2. Click the create new layer icon, name your layer and change the colour and line type.
3. Double click on that layer to draw on that layer, a green check mark will show you what layer you are on.
4. If you click on the sun icon in the Layer Properties Manager, it will turn into a snowflake which means you have frozen your layer (cannot see it).

Viewing and Changing Layers



The drop-down menu on the layer panel will also show you what layer you are working in. You can change what layer you are working in using the drop down menu (click on desired layer) and you can freeze and unfreeze layers by clicking on the sun or snowflake.

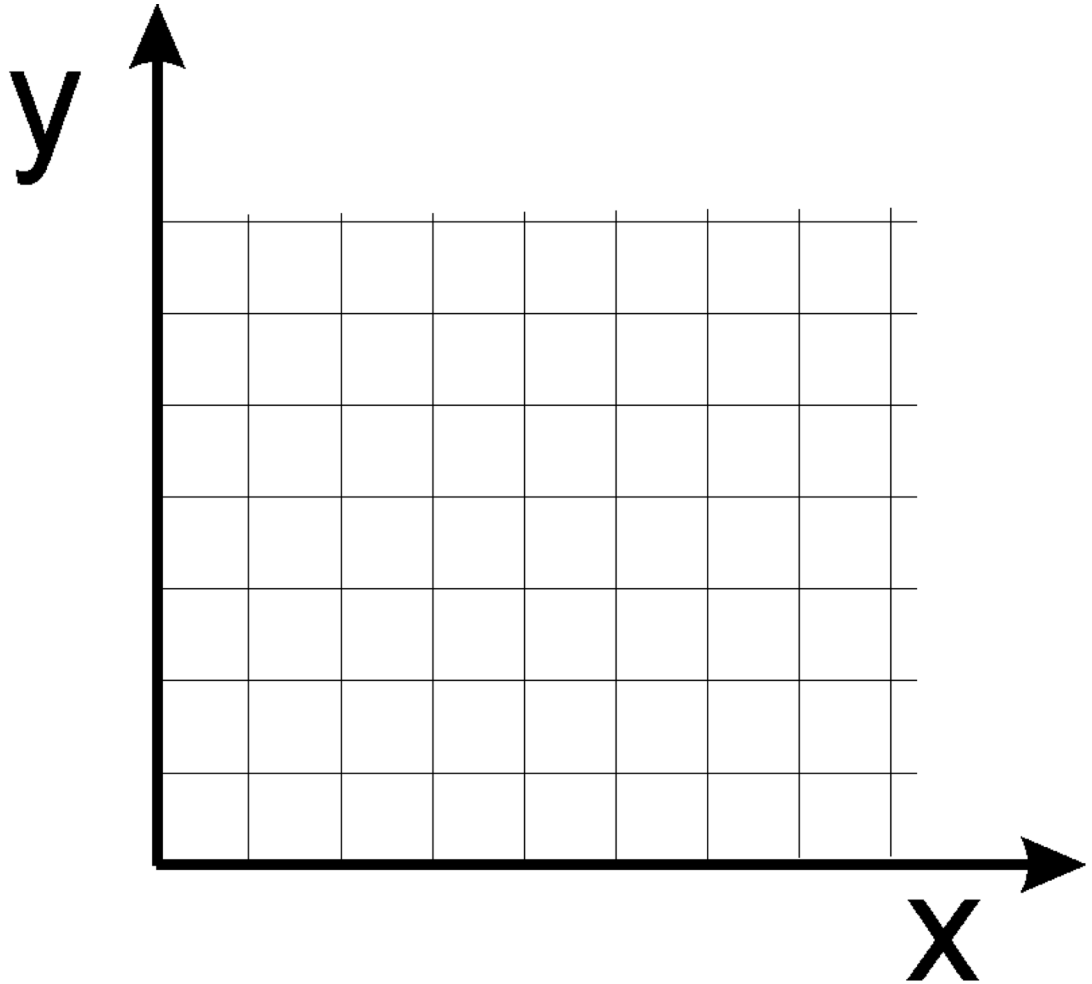
Viewing and Changing Layers in Properties



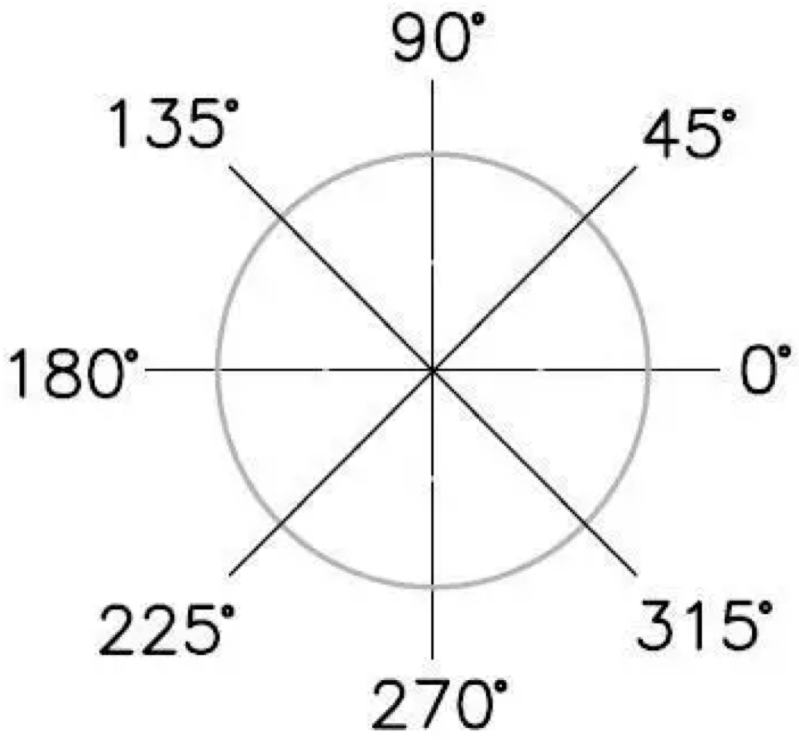
You can also see what layer a line is on or change the layer of a line by going into properties. Type “PROPERTIES” into the command bar and press enter. Click on the line you want to change the layer of. Click on the layer (in the properties window) and a drop-down menu will appear. Here you can change what layer the line is on by clicking on your desired layer.

Drawing lines

- There are two basic ways to draw lines on AutoCAD, using absolute or relative coordinates.
- Drawing lines using absolute coordinates means you first specify a starting (x,y) coordinate, then you specify an end (x,y) coordinate
- Drawing lines using relative coordinates means you first establish a starting coordinate (click on another line or type in a coordinate) then you specify how *long* you want the line to be and what *direction* it goes in (given by an angle)
- The format for this is **@length<angle** when you type it in on the command bar

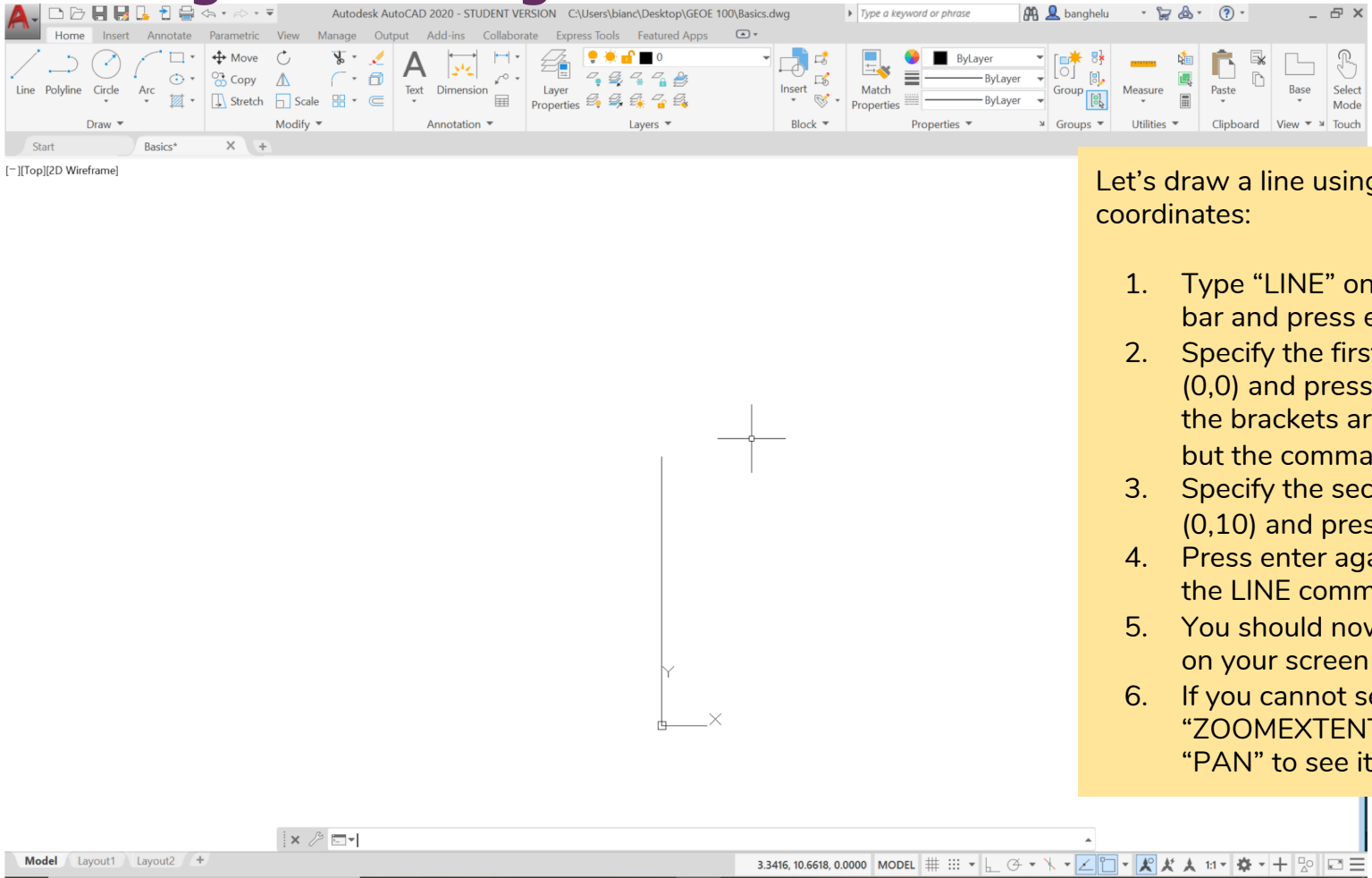


Use positive (x,y) coordinates when drawing lines on AutoCAD such as (20,30). It can be helpful to start a drawing at (0,0) and draw using relative coordinates after that, rather than using absolute coordinates.



AutoCAD recognizes angles in this format, with 0 degrees starting on the right hand side and angles increasing counterclockwise from 0. You can tell AutoCAD what direction you want a line to go in by using angles. You can also input negative angles into AutoCAD (if you have learned them in school).

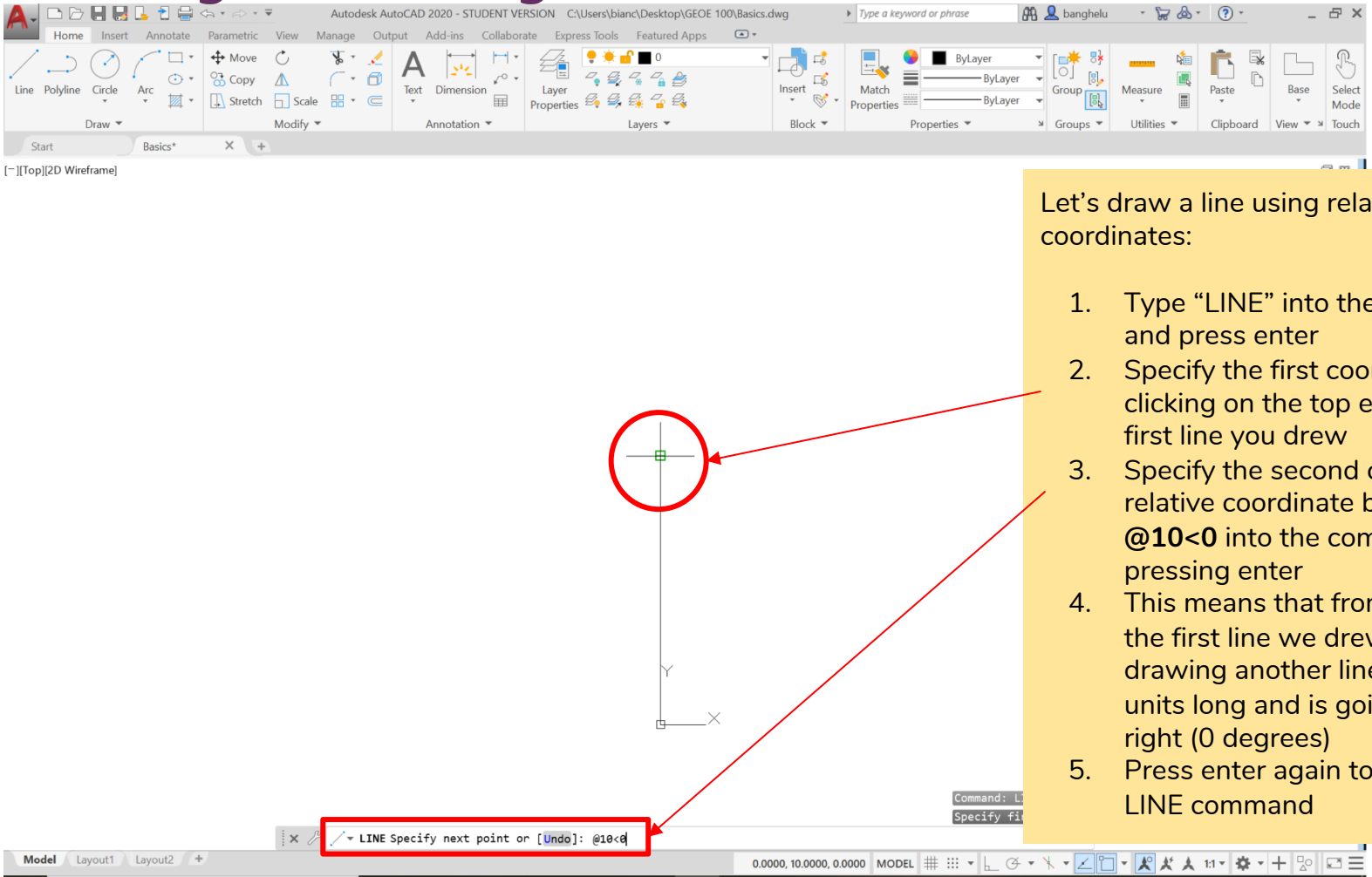
Drawing Lines Using Absolute Coordinates



Let's draw a line using absolute coordinates:

1. Type "LINE" on the command bar and press enter
2. Specify the first coordinate as (0,0) and press enter (*note, the brackets are not needed but the comma is)
3. Specify the second point as (0,10) and press enter
4. Press enter again to terminate the LINE command
5. You should now have this line on your screen
6. If you cannot see it perform the "ZOOMEXTENTS" command or "PAN" to see it

Drawing Lines Using Relative Coordinates



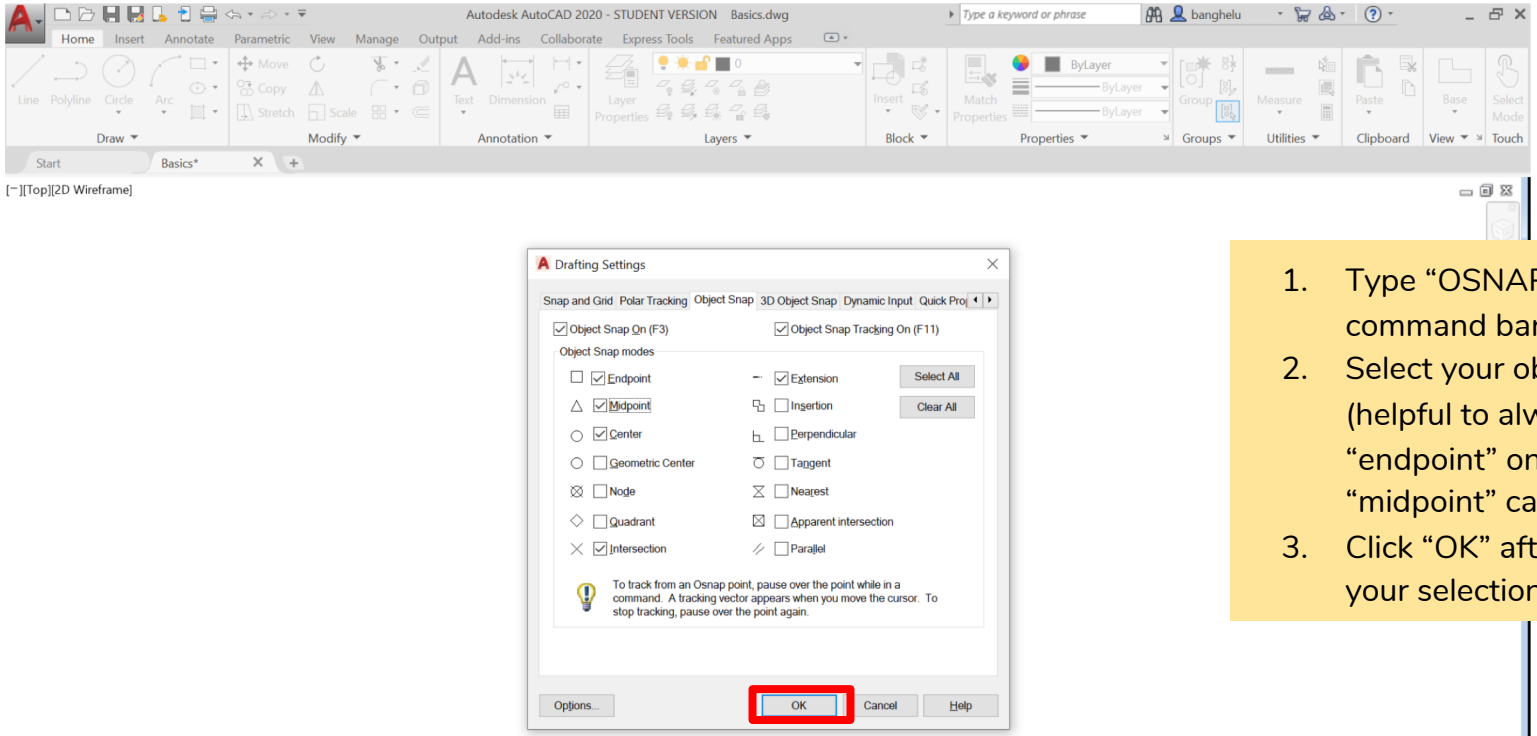
Let's draw a line using relative coordinates:

1. Type "LINE" into the command bar and press enter
2. Specify the first coordinate by clicking on the top endpoint of the first line you drew
3. Specify the second coordinate as a relative coordinate by typing **@10<0** into the command bar and pressing enter
4. This means that from the top of the first line we drew, we are drawing another line that is 10 units long and is going directly right (0 degrees)
5. Press enter again to terminate the LINE command

OSNAP

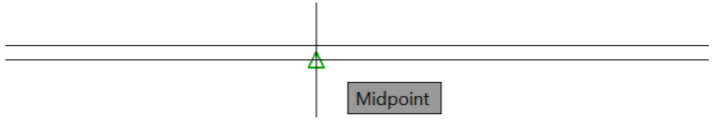
- The Object Snap command opens Drafting Settings.
- This lets you choose what your pointer can “snap” onto such as the end or middle of a line, the intersection between two lines, or the centre of a circle.
- “Snapping” onto a line means that you can click onto different parts of it and draw new lines off of it.
- Turning these snap settings on and off lets you control what you can click on and draw lines off of.

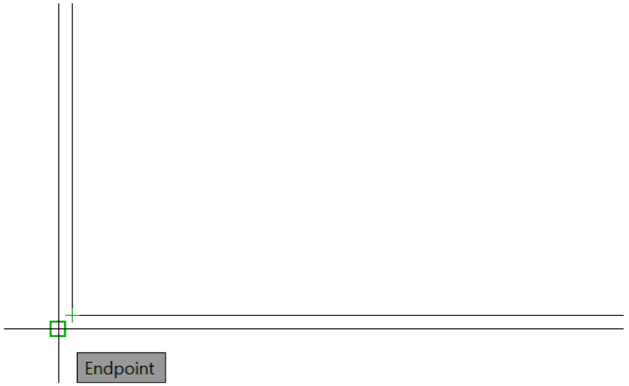
Drafting Settings

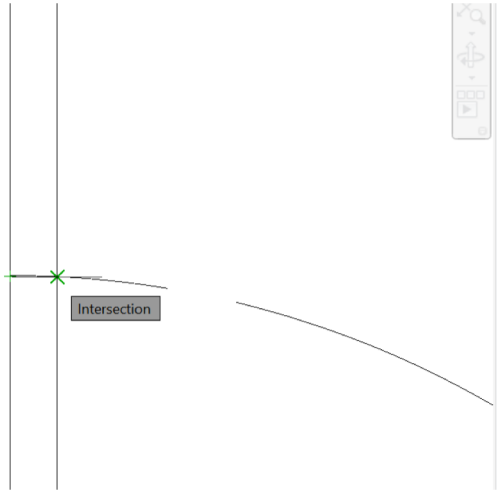


1. Type “OSNAP” into the command bar and press enter
2. Select your object snap modes (helpful to always have “endpoint” on, “centre” and “midpoint” can also be helpful)
3. Click “OK” after you have made your selections

Snap Options (What Snapping Looks Like)





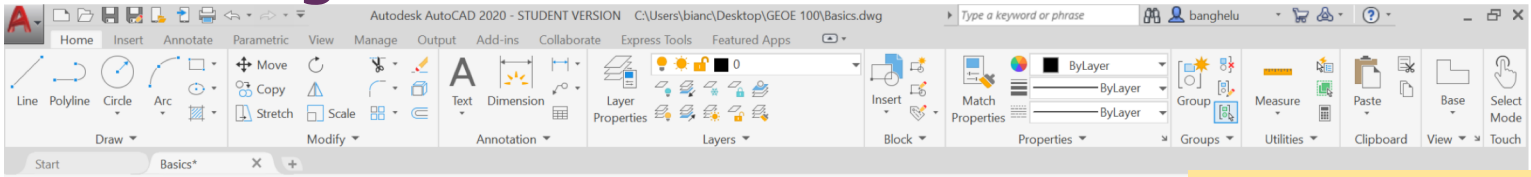


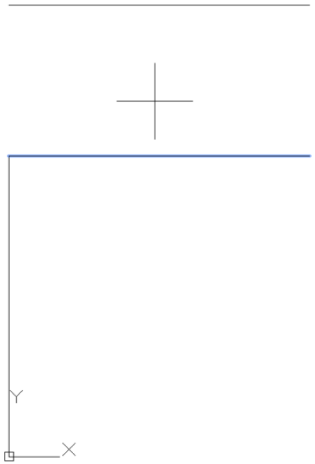
As mentioned before, OSNAP settings determine what you can “snap” onto when drawing lines. For example, if you did not have the OSNAP setting for “Midpoint” turned on, you would not be able to click onto the midpoint of the line as is shown in the picture.

OFFSET

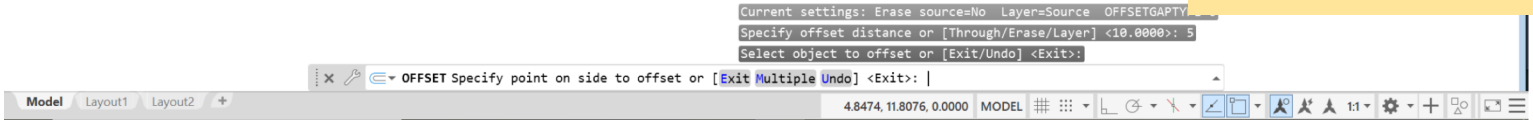
- The OFFSET command creates parallel lines a specific distance apart.
- You can specify what objects you want to make a parallel for, and what distance away from the original you want the parallel object to be.
- You can offset straight lines, curved lines, and circles.

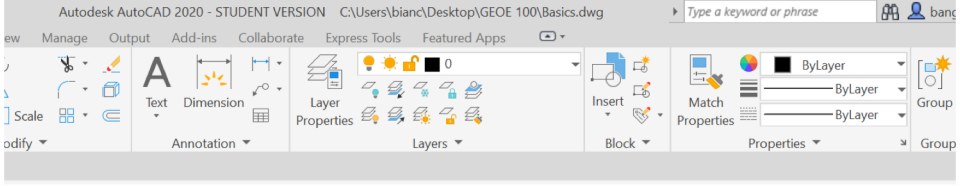
Offsetting Lines

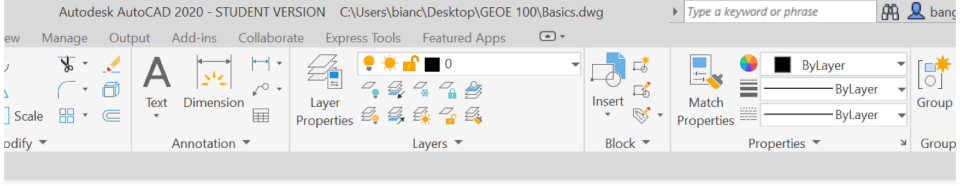




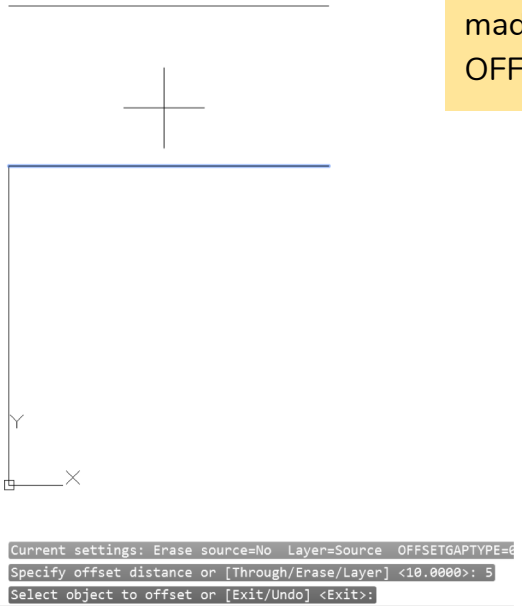
1. Type “OFFSET” into the command bar and press enter.
2. Specify the offset distance by entering in a numeric value and press enter.
3. Select the object to offset by clicking on it.
4. Specify the direction of the offset by moving your pointer to one side of the pre-existing line, hover your mouse to see where the new line will go, click when you want to finalize the position.
5. Press “esc” to terminate the command.

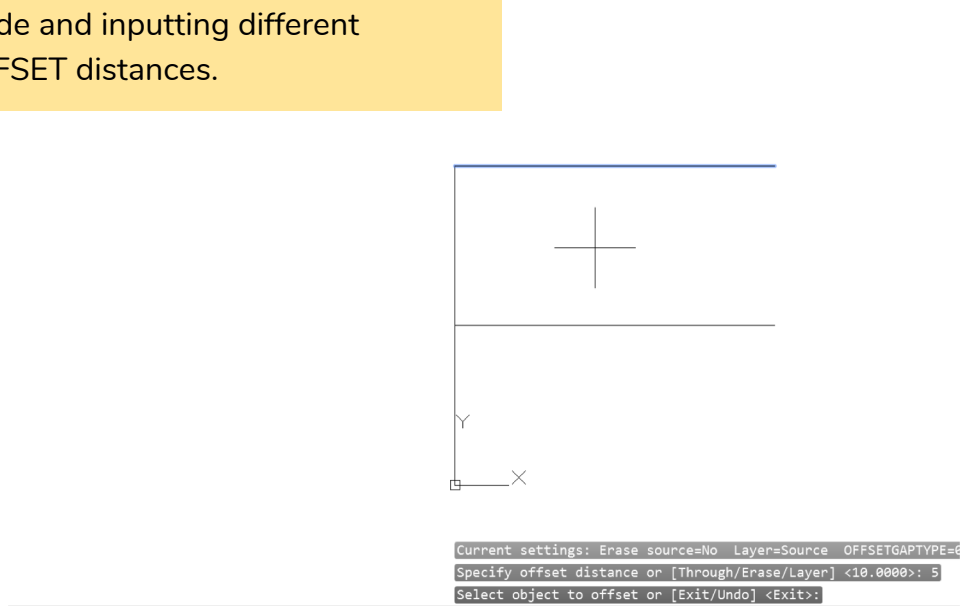






Experiment with the OFFSET command by hovering your cursor above and below the line you initially made and inputting different OFFSET distances.

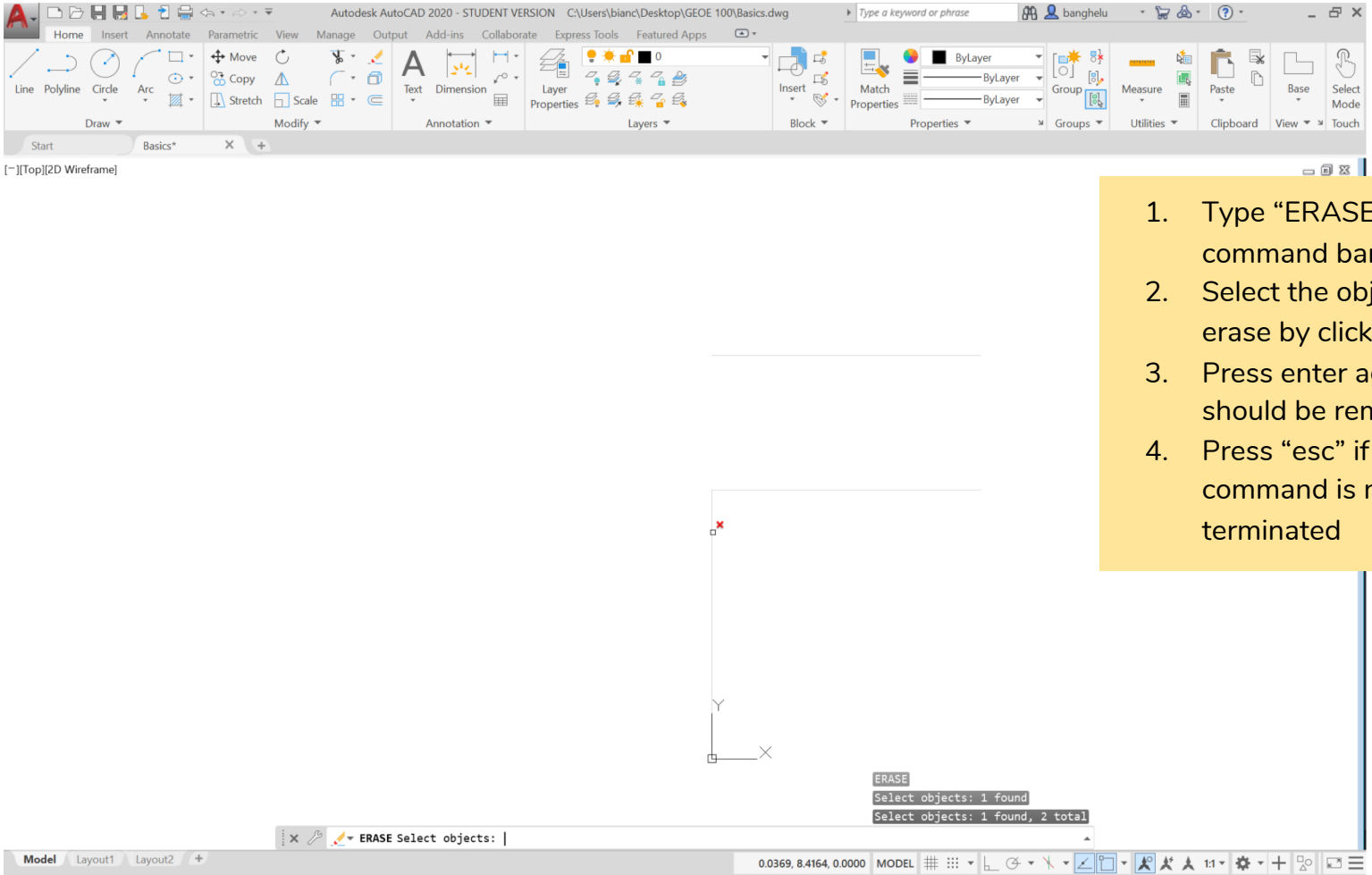




ERASE

- Erase lines, circles, or curves you have drawn
- The ERASE command always erases the entire line and not just portions of it

Erasing Lines

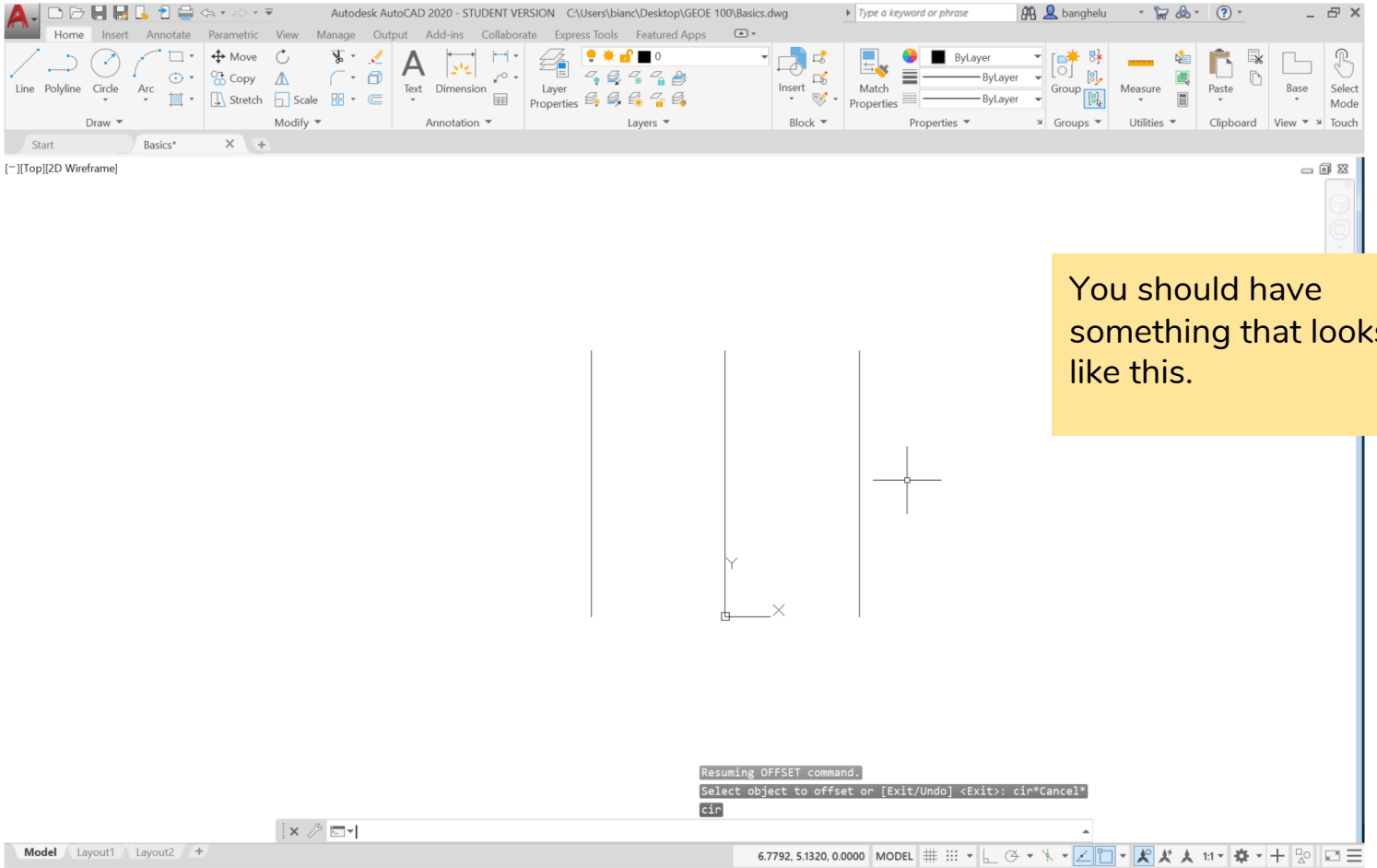


1. Type “ERASE” into the command bar and press enter.
2. Select the objects you want to erase by clicking on them.
3. Press enter again and the lines should be removed.
4. Press “esc” if the ERASE command is not automatically terminated

Try This!

- Draw a line starting at (0,0) going to (0,10)
- OFFSET the centre line 2 units to the left
- OFFSET the centre line 2 units to the right

Try this example

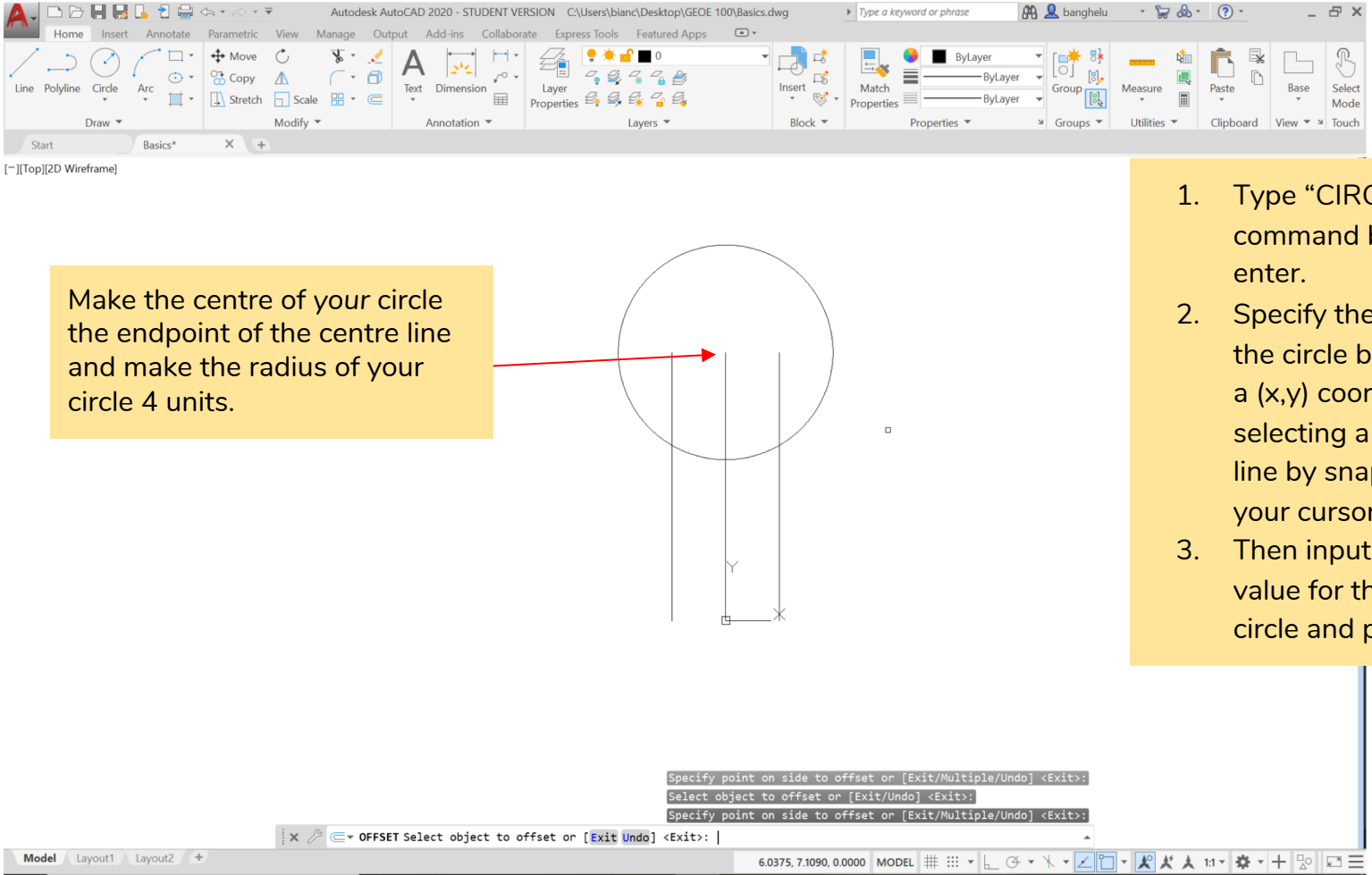


You should have something that looks like this.

CIRCLE

- Draw circles of different sizes

Drawing a Circle



Make the centre of your circle the endpoint of the centre line and make the radius of your circle 4 units.

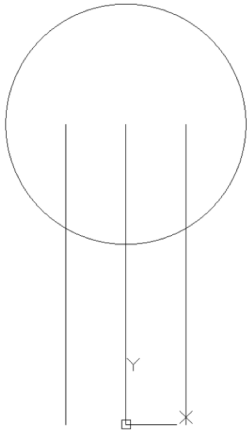
1. Type “CIRCLE” into the command bar and press enter.
2. Specify the centre point of the circle by either typing in a (x,y) coordinate or selecting a point on another line by snapping onto it with your cursor.
3. Then input a numerical value for the radius of your circle and press enter.

FILLET

- Make a round (fillet) between two objects.

Drawing a Fillet

Input a value of 2 units for the radius of your fillet.



1. Type “FILLET” into the command bar and press enter.
2. Type “r” and press enter (we want to make a round between two objects so we need to input a radius).
3. Input a numerical value for the radius of the round and press enter.

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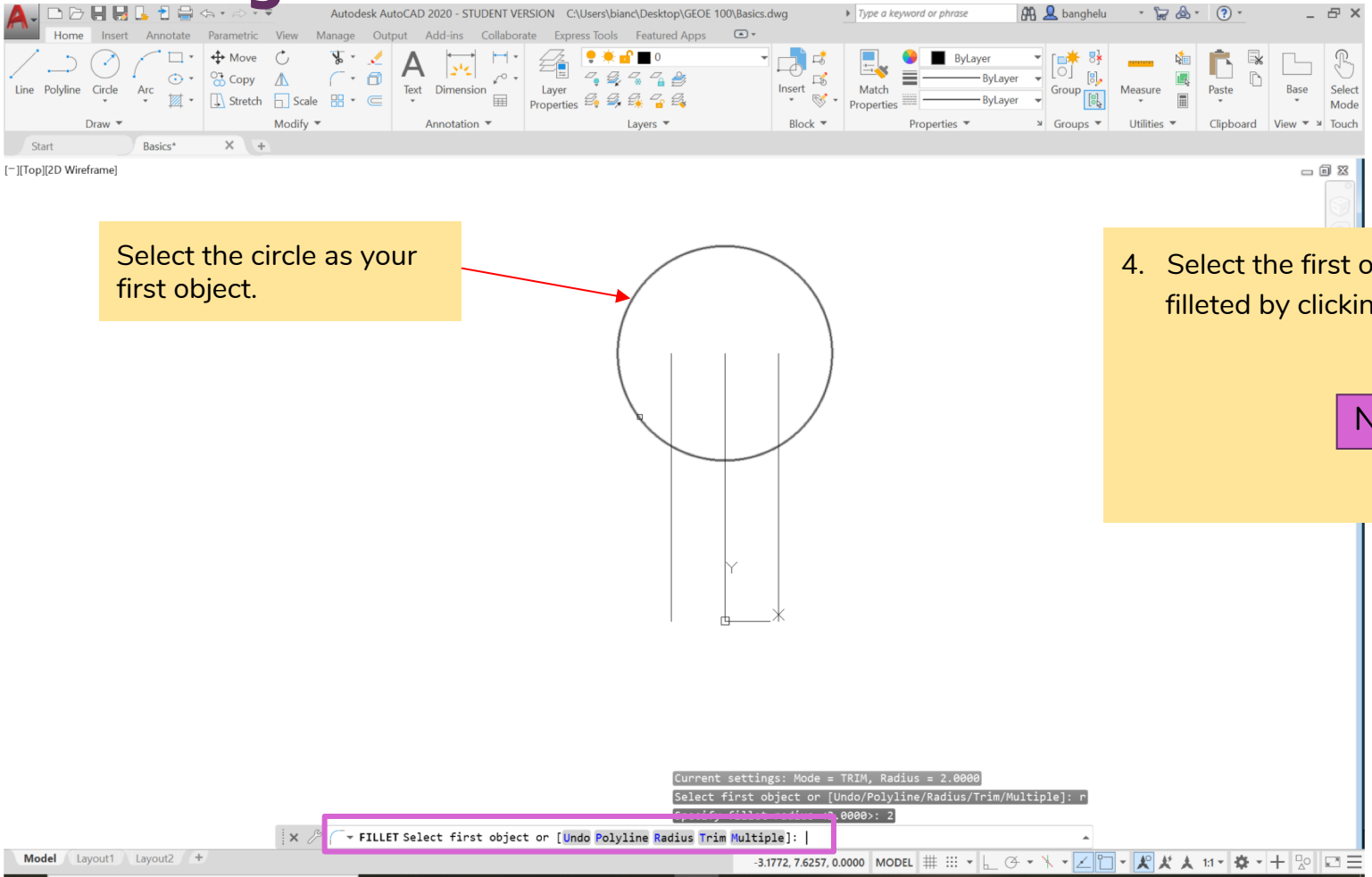
Drawing a Fillet



4. Select the first object to be filleted by clicking on it.

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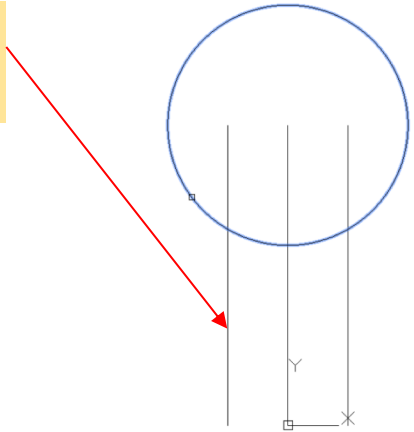
Drawing a Fillet

Select the left line as your second object.



5. Select the second object by clicking on it and a round will be made between the two.

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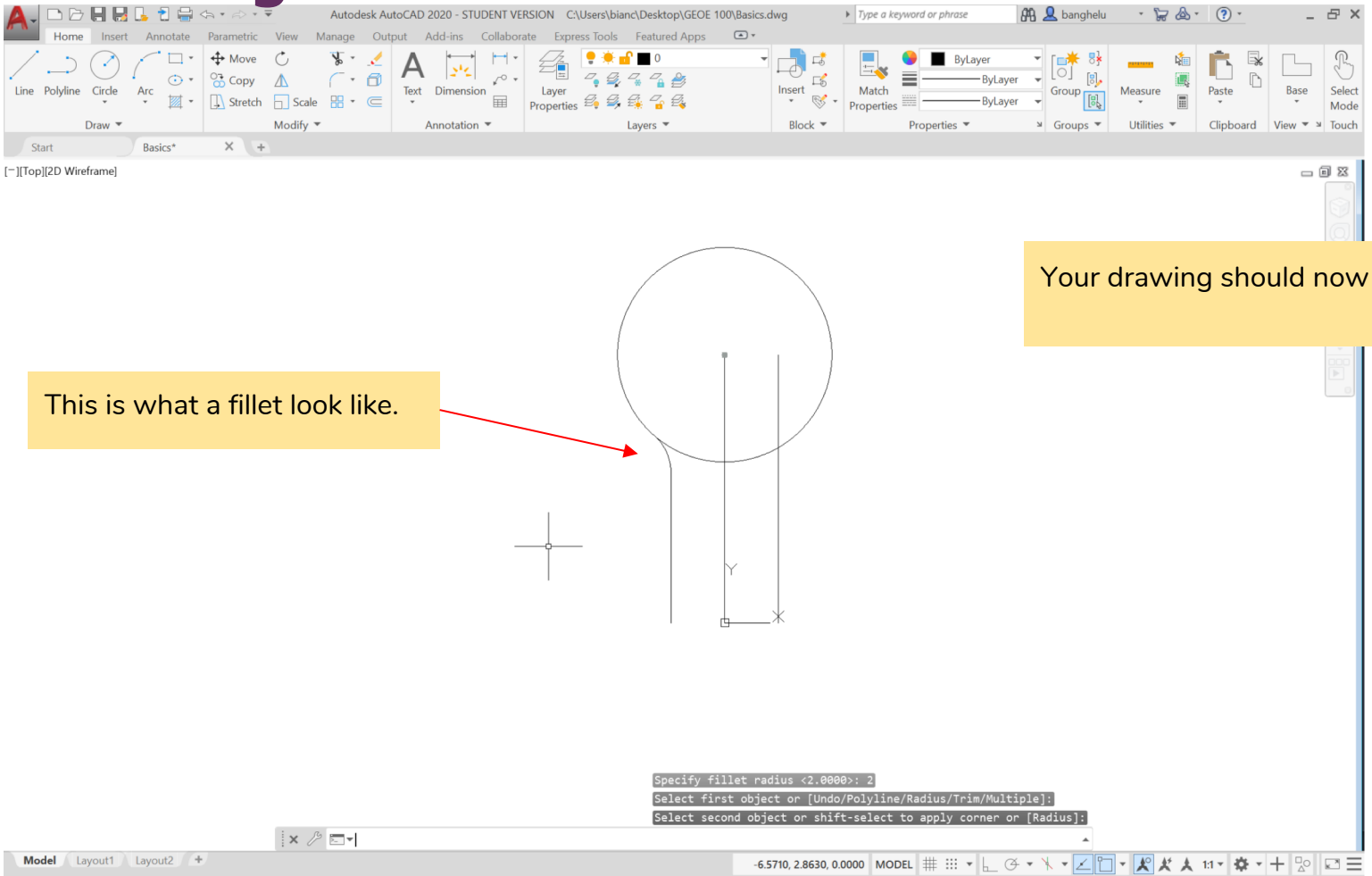
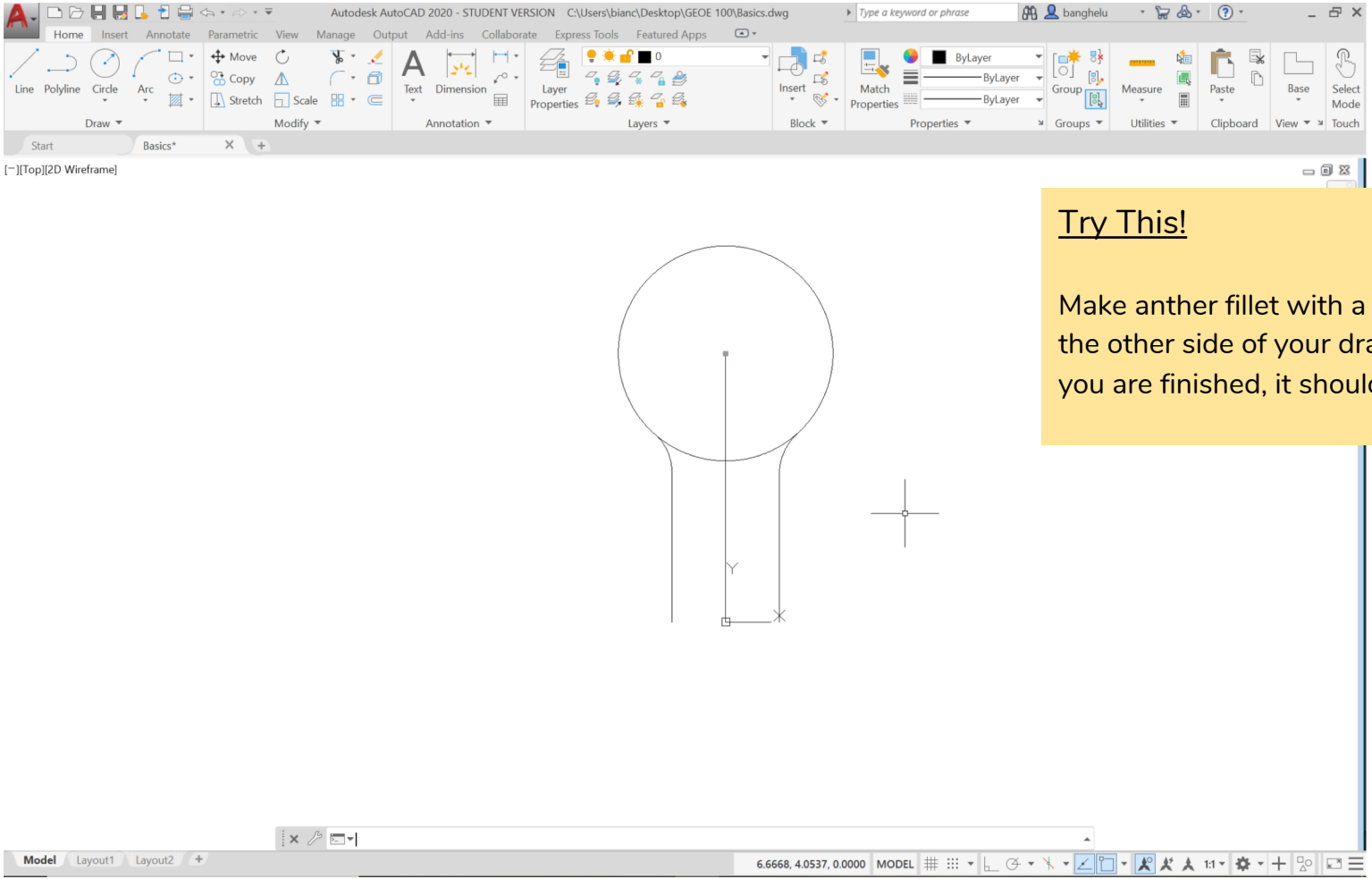
Drawing a Fillet



This is what a fillet look like.

Your drawing should now look like this.



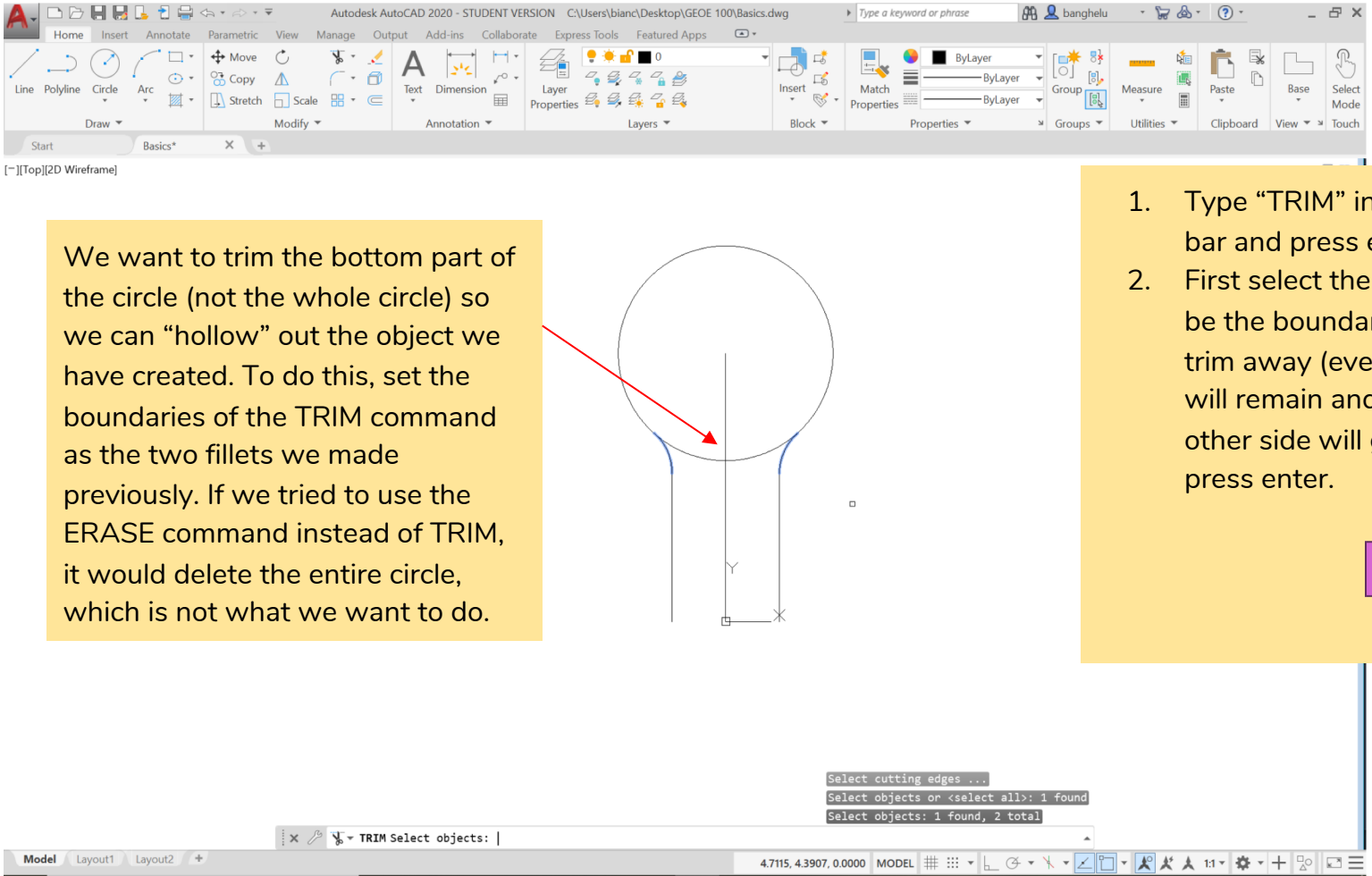
Try This!

Make anther fillet with a radius of 2 on the other side of your drawing. Once you are finished, it should look like this.

TRIM

- TRIM removes portions of your linework without deleting the entire line
- When using the TRIM command, you cannot start deleting sections right away
- You must first set “boundaries” which tell AutoCAD that you want to delete the portion of the drawing that is between the two lines you set as the “boundaries”

Trimming a Line



We want to trim the bottom part of the circle (not the whole circle) so we can "hollow" out the object we have created. To do this, set the boundaries of the TRIM command as the two fillets we made previously. If we tried to use the ERASE command instead of TRIM, it would delete the entire circle, which is not what we want to do.

1. Type "TRIM" into the command bar and press enter.
2. First select the linework that will be the boundaries of what you trim away (everything on one side will remain and everything on the other side will get erased) and press enter.

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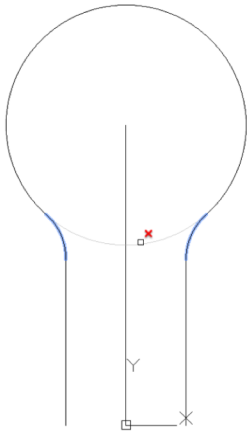
Trimming a Line

Click on the bottom of the circle between the boundaries we set to trim it.



3. Click on the line you want to trim
4. Press enter to terminate the TRIM command.

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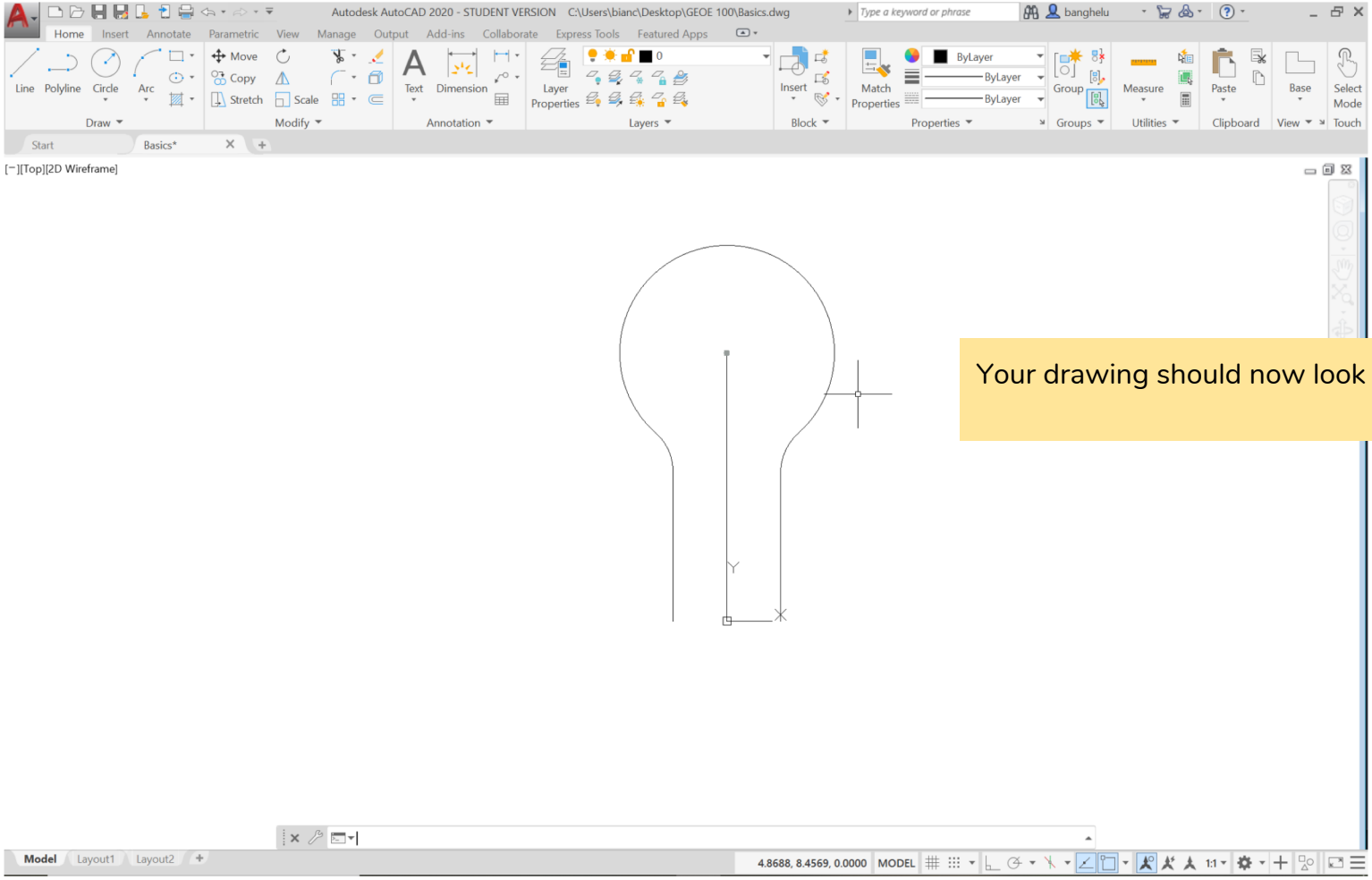
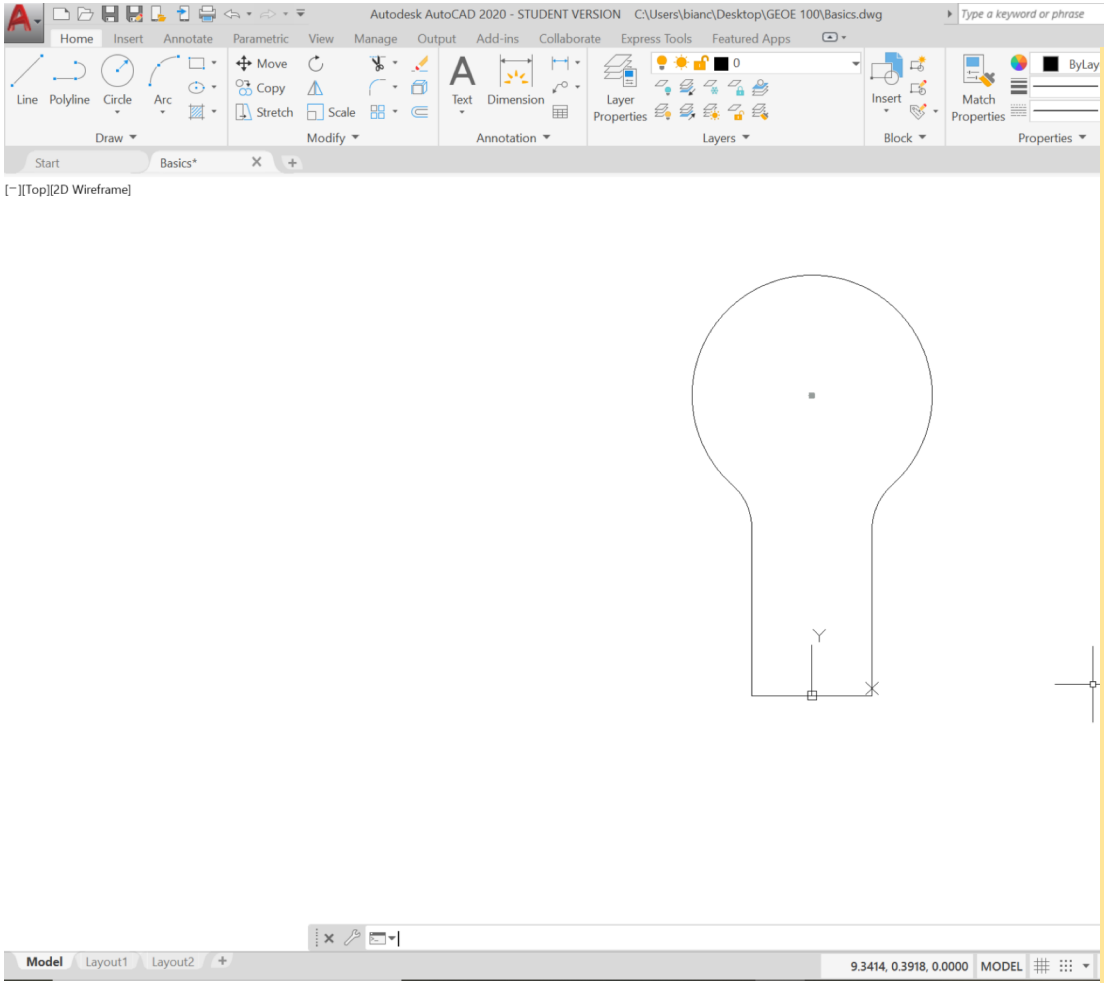


Your drawing should now look like this.

Try This!

- ERASE the centre line of your drawing
- Make your drawing an enclosed object (so that we can fill it in with colour later)
- Enclose your object by drawing a horizontal LINE at the bottom of the object
- Go to the next slide to see a picture of the final result



To draw the horizontal line:

1. Type “LINE” into the command bar and press enter
2. Specify the first coordinate as the endpoint of the left line by clicking on it
3. Specify the second coordinate as the endpoint of the right line by clicking on it
4. Press enter to terminate the LINE command

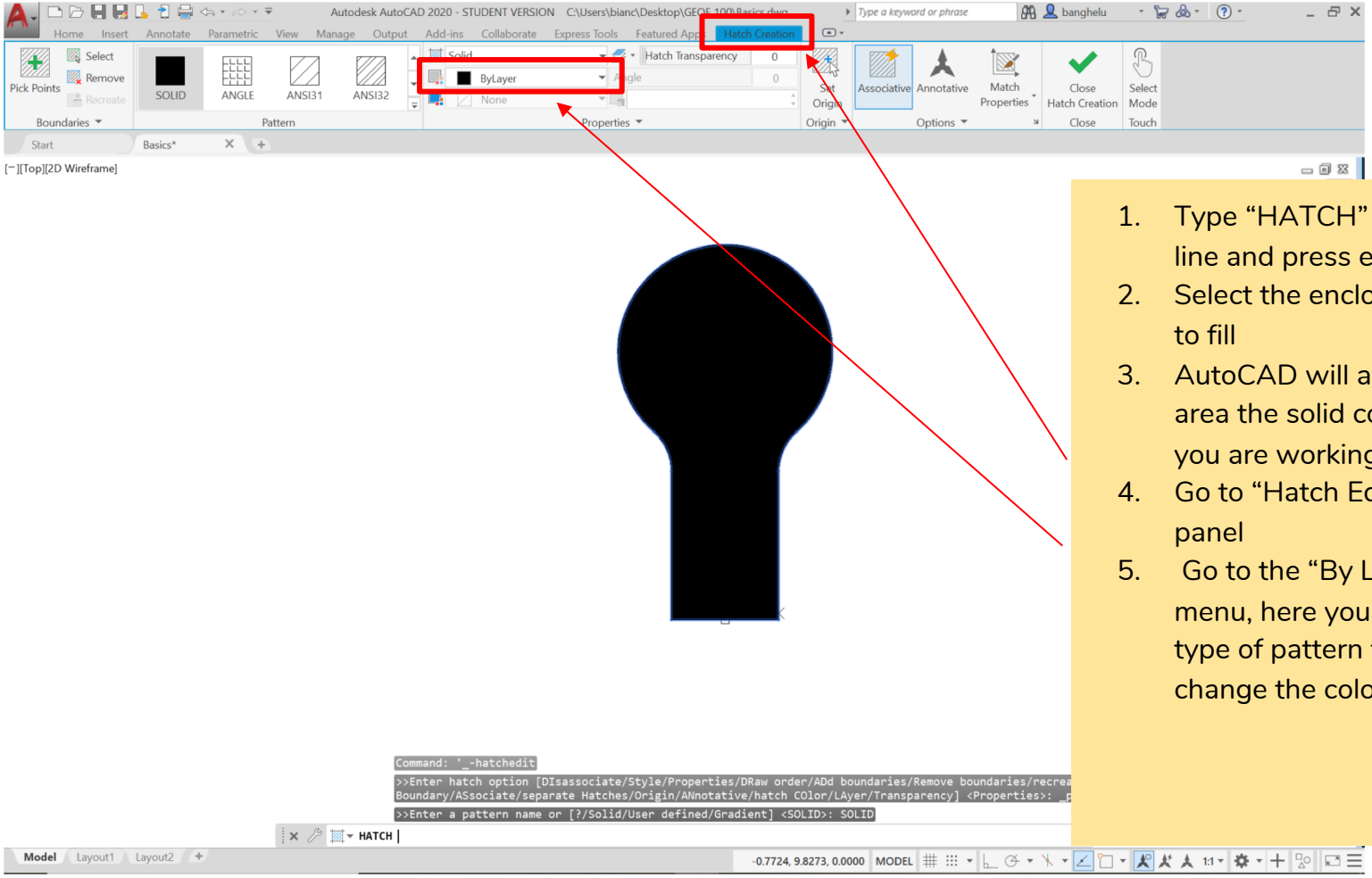
OR

1. Type “LINE” into the command bar and press enter
2. Specify the first coordinate as the endpoint of the left line by clicking on it
3. Specify the second coordinate using a distance and angle, type **@4<0** and press enter
4. Press enter to terminate the LINE command
5. If you had specified your first coordinate as the right line, the relative coordinate would have been **@4<180** (so the line is drawn to the left, 180 degrees)

HATCH

- Add texture, patterns, or fill enclosed objects with solid or gradient colour

Filling an Enclosed Object Using HATCH



1. Type “HATCH” into the command line and press enter
2. Select the enclosed area you want to fill
3. AutoCAD will automatically fill the area the solid colour of the layer you are working on
4. Go to “Hatch Editor” on the top panel
5. Go to the “By Layer” drop-down menu, here you can change the type of pattern for your fill and change the colour of the fill.

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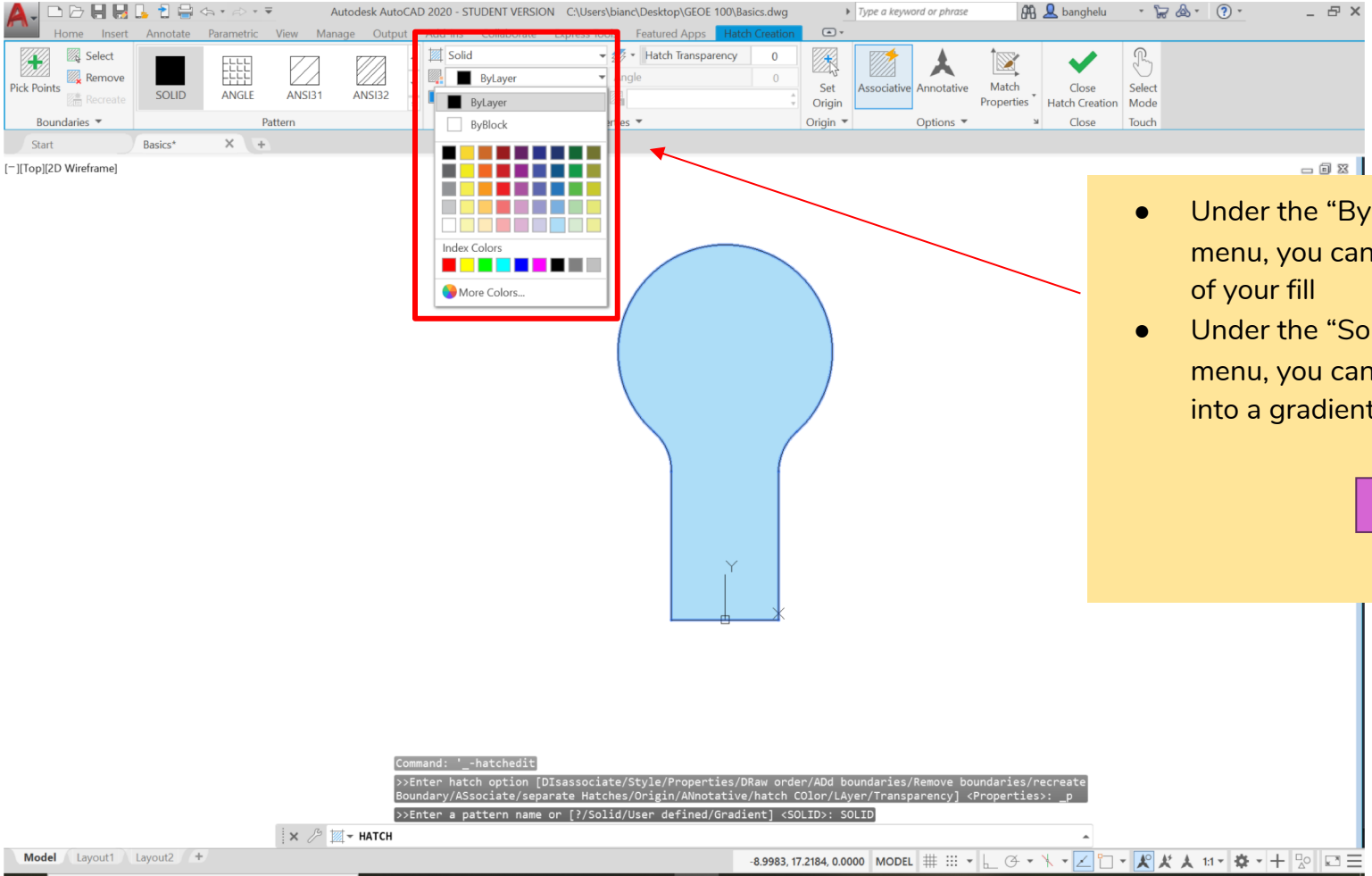
Changing the Fill Colour: Hatch Editor



- Under the “By Layer” drop-down menu, you can change the colour of your fill
- Under the “Solid” drop-down menu, you can make your solid fill into a gradient colour

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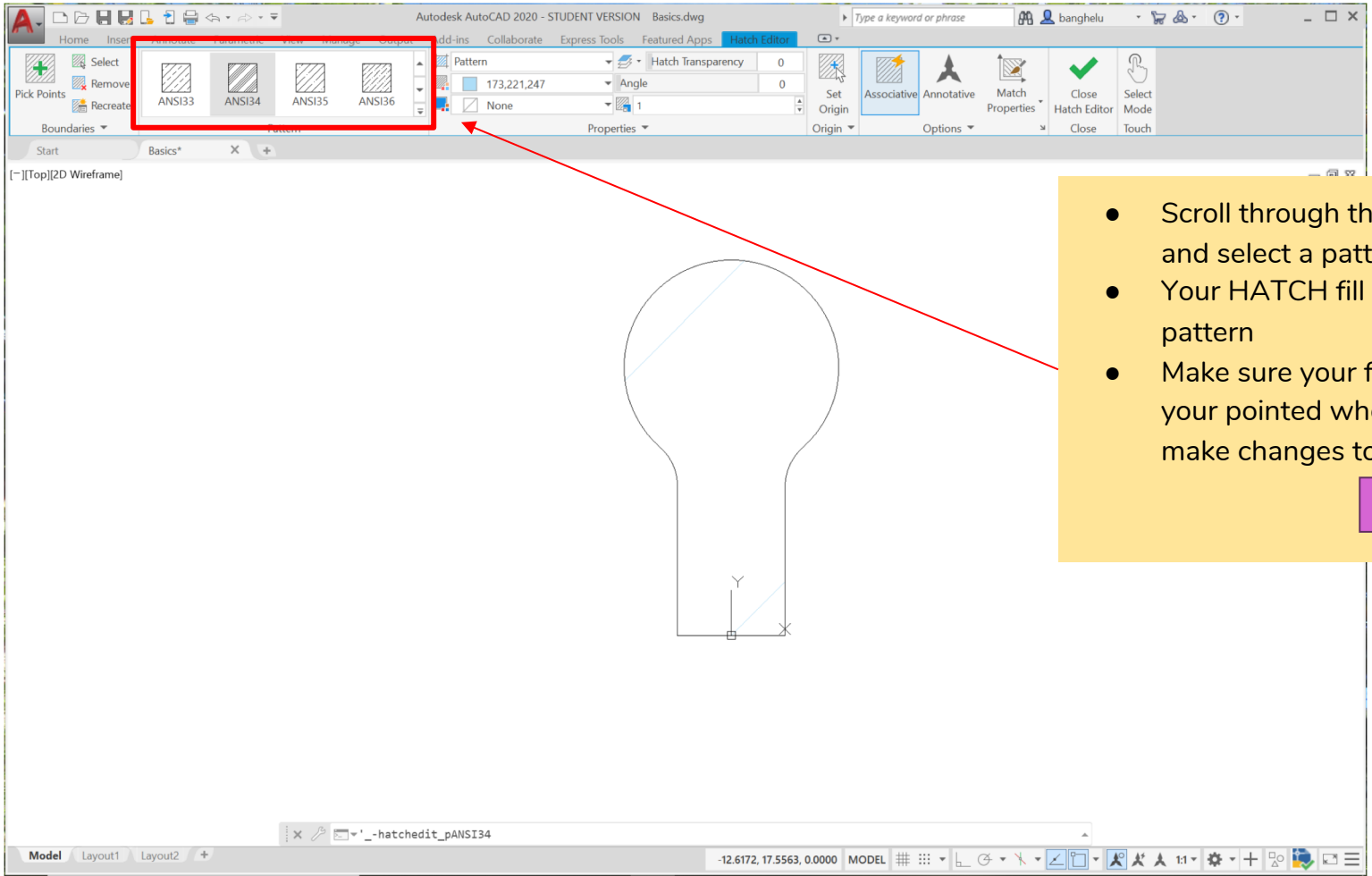
Changing the Hatch Pattern: Hatch Editor



- Scroll through the pattern options and select a pattern
- Your HATCH fill will change to this pattern
- Make sure your fill is selected with your pointed when you want to make changes to it

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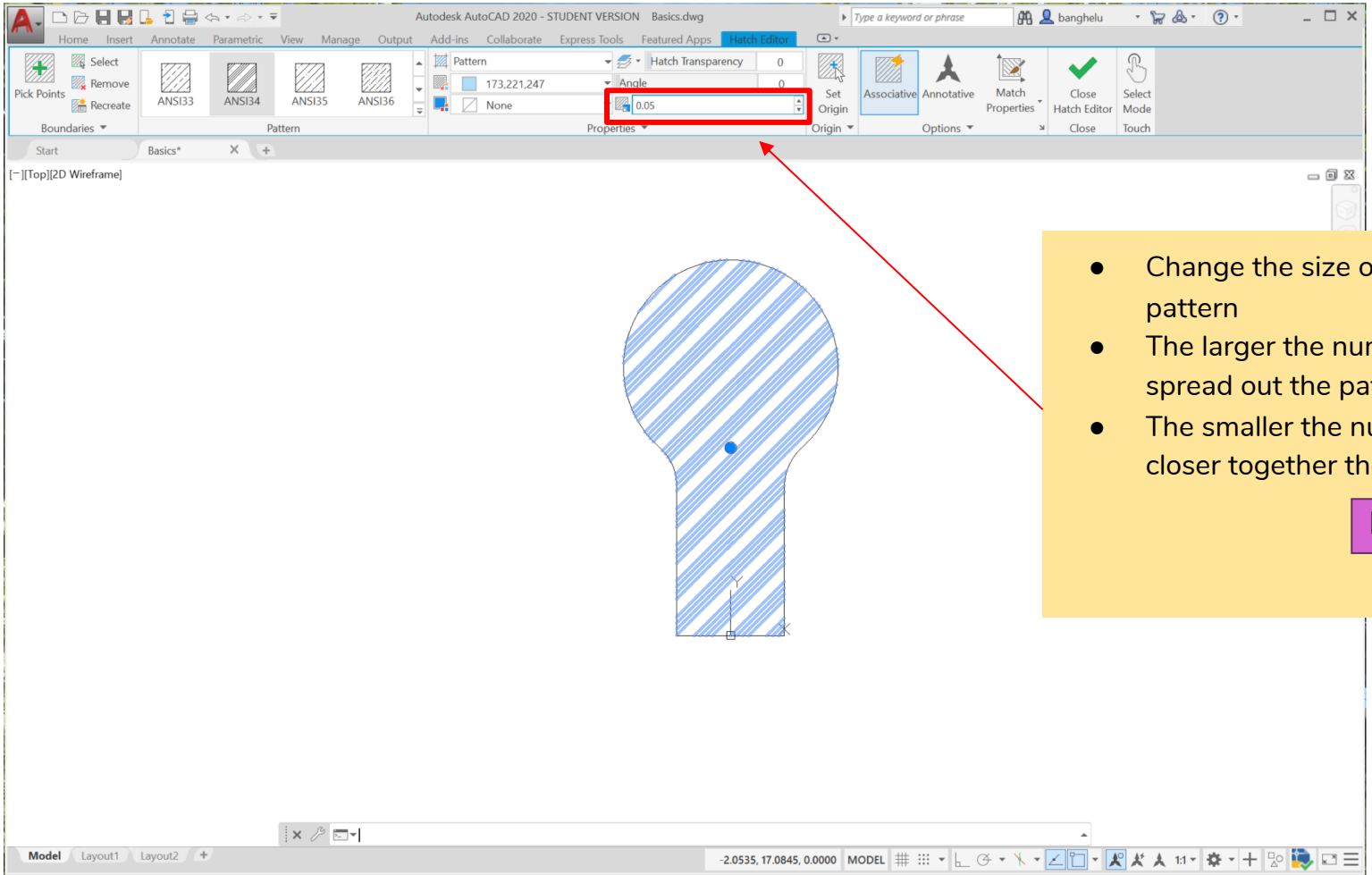
Changing the Hatch Pattern Size: Hatch Editor



- Change the size of your HATCH pattern
- The larger the number, the more spread out the pattern will be
- The smaller the number, the closer together the pattern will be

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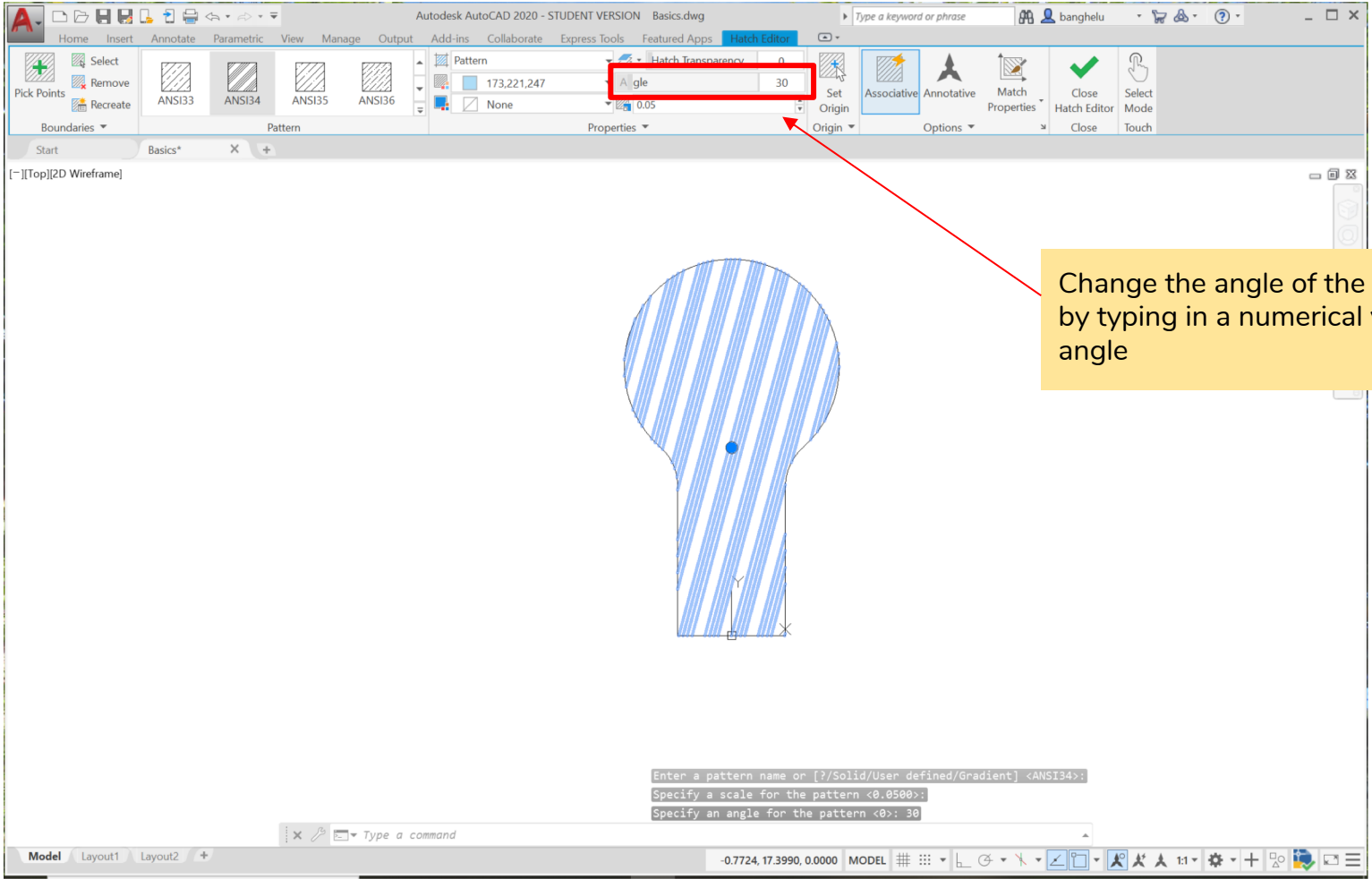
Changing the Hatch Pattern Angle: Hatch Editor

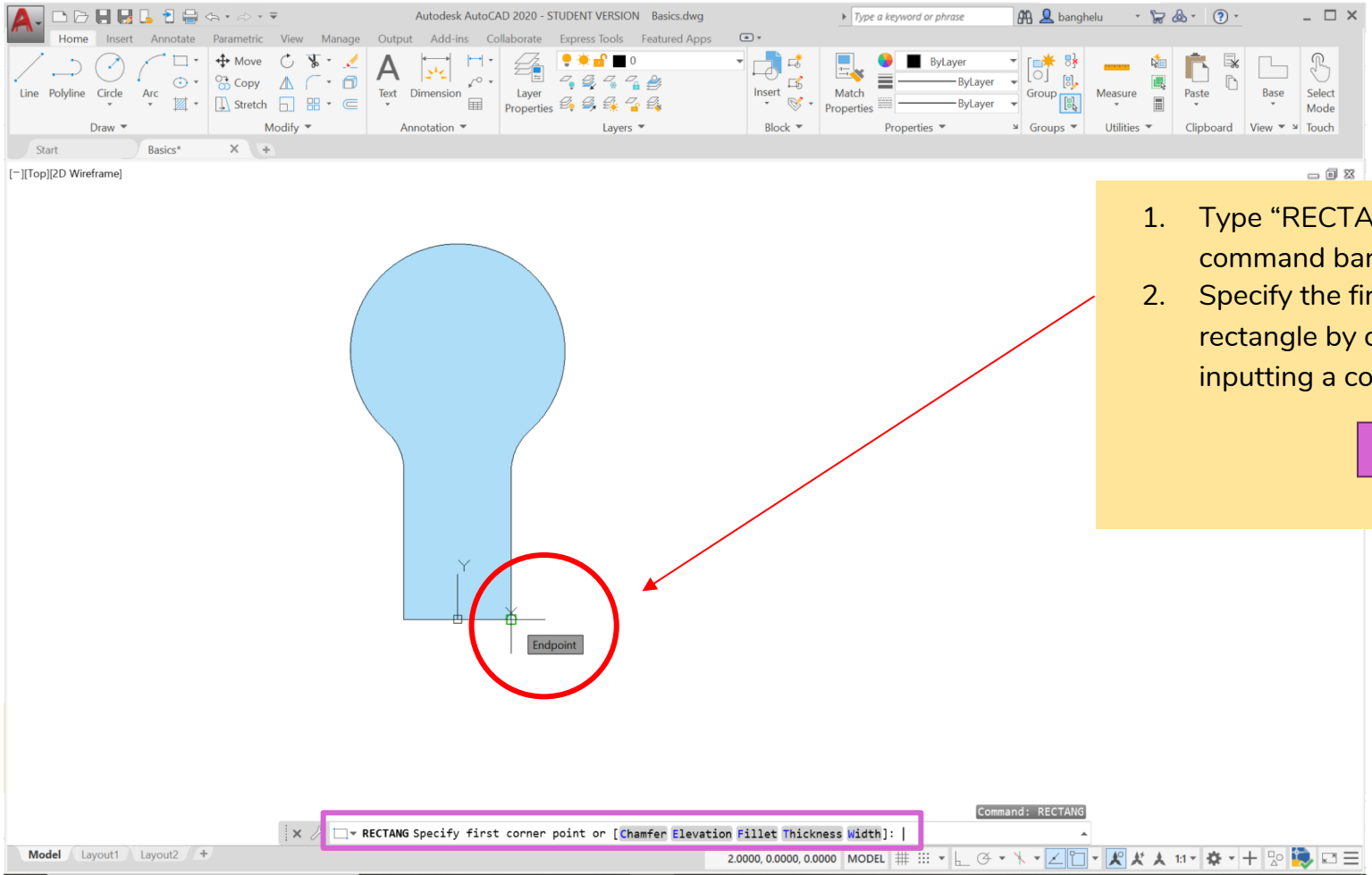


Change the angle of the HATCH pattern by typing in a numerical value for the angle

RECTANG

- Draw rectangles of different sizes

Drawing a Rectangle



1. Type “RECTANG” into the command bar and press enter.
2. Specify the first corner of the rectangle by clicking on a line or inputting a coordinate.

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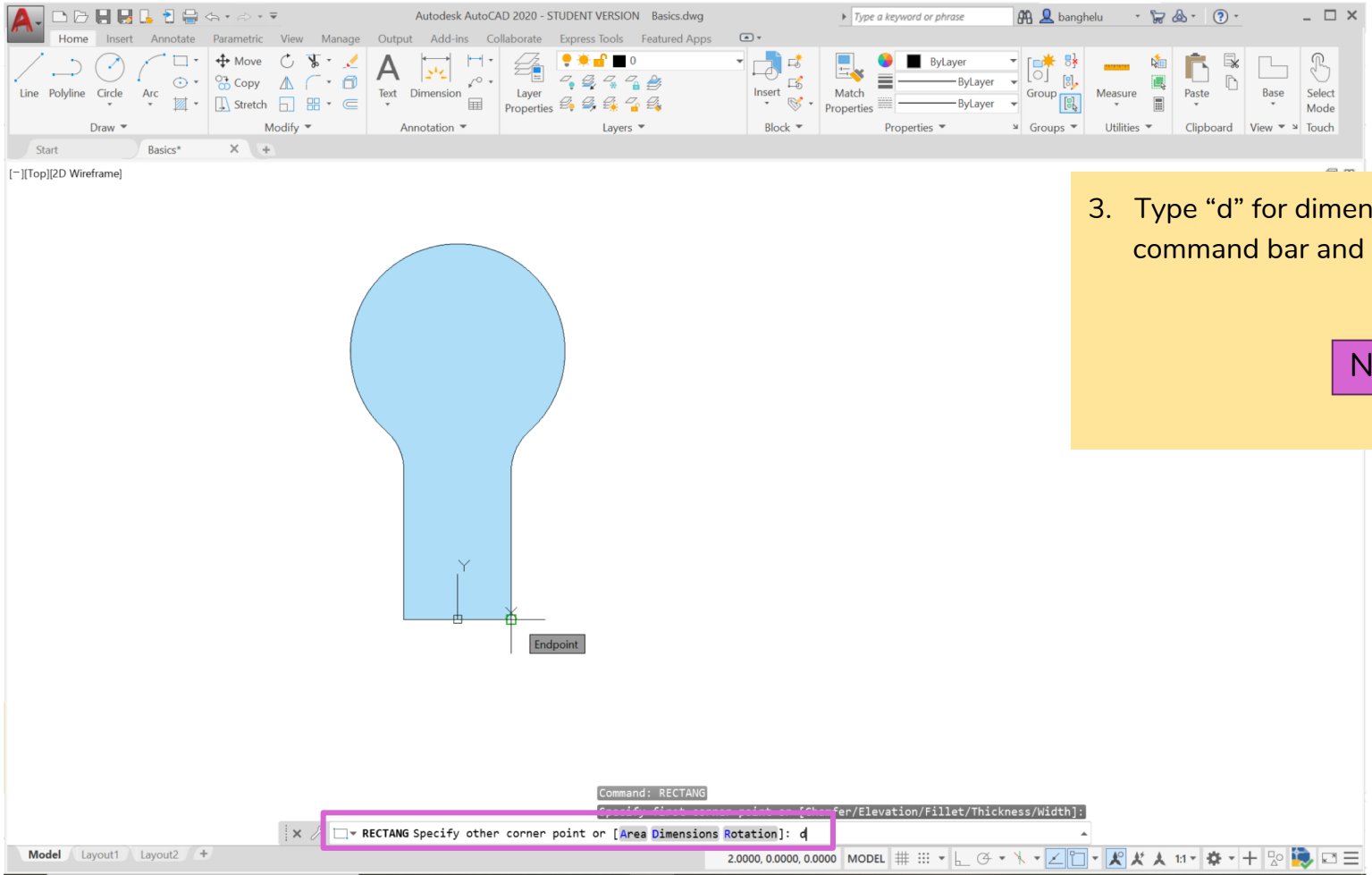
Drawing a Rectangle



3. Type “d” for dimensions into the command bar and press enter.

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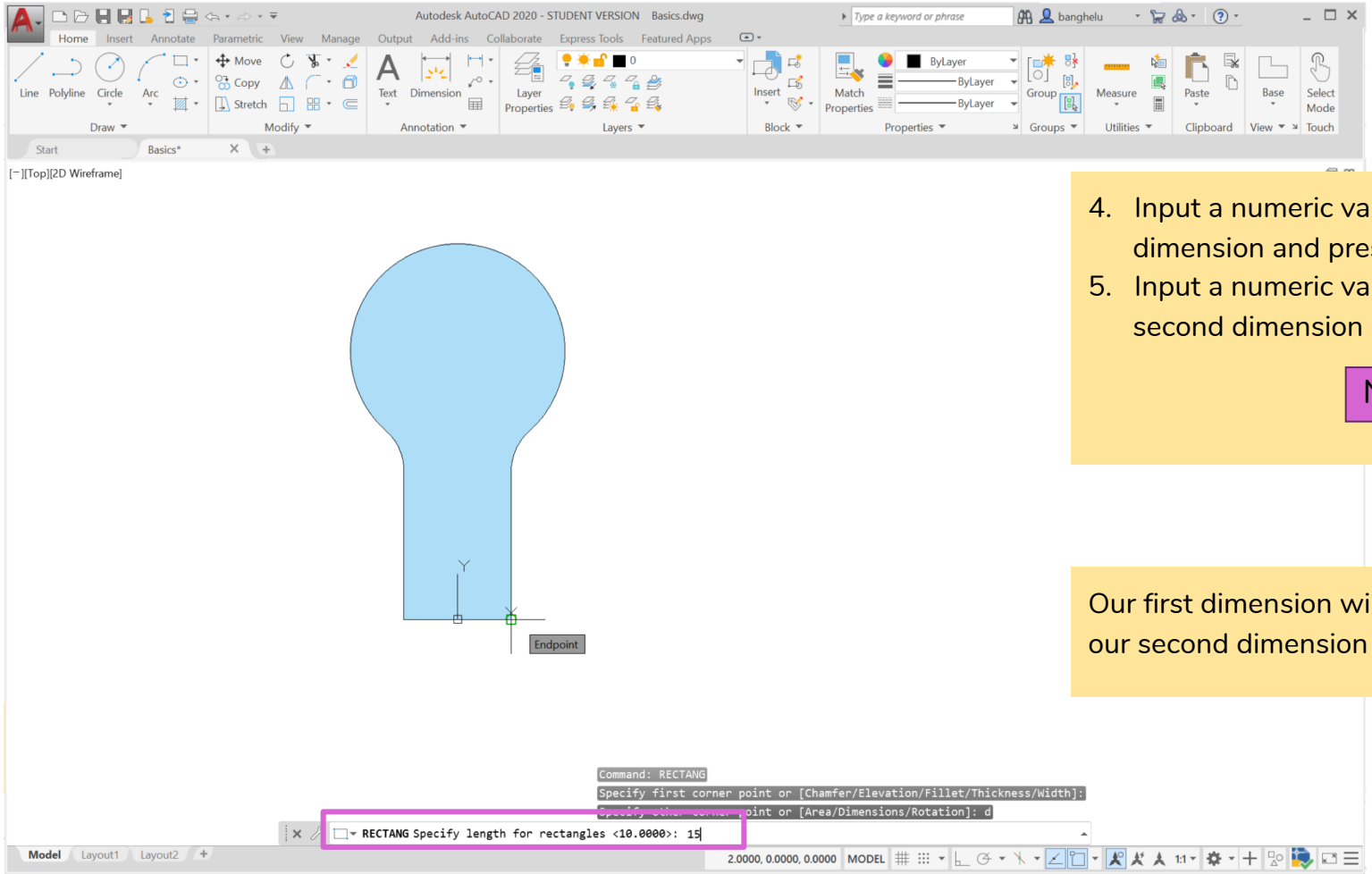
Drawing a Rectangle



4. Input a numeric value for the first dimension and press enter.
5. Input a numeric value for the second dimension and press enter.

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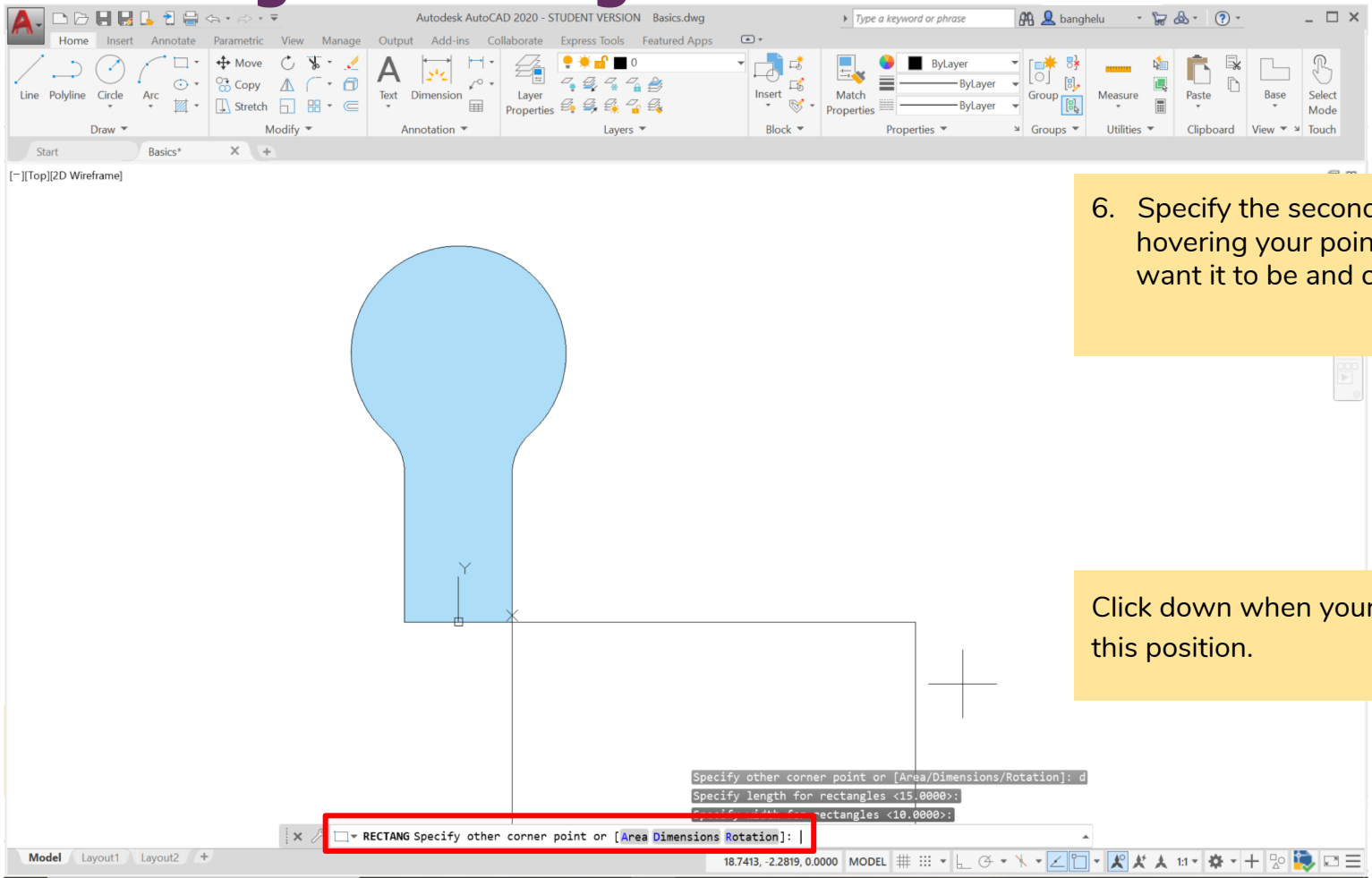
Our first dimension will be 15 units, our second dimension will be 10 units.

Drawing a Rectangle



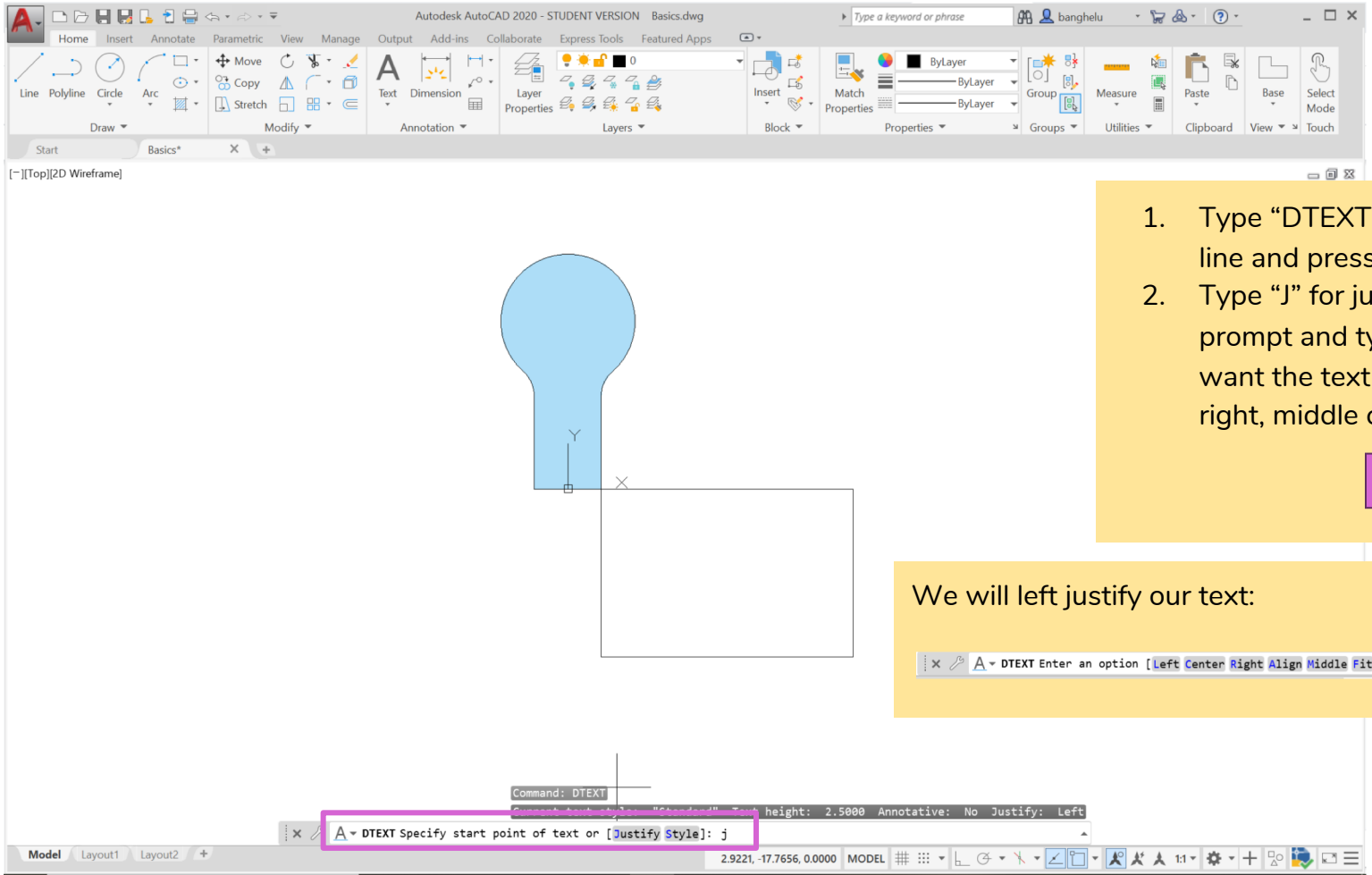
6. Specify the second corner by hovering your pointer where you want it to be and clicking down.

Click down when your rectangle is in this position.

DTEXT

- Write a single line of text
- Decide on the text justification (if the text is aligned on the right side, left side, or center) and text height
- DTEXT stands for Dynamic Text

Writing a Single Line of Text

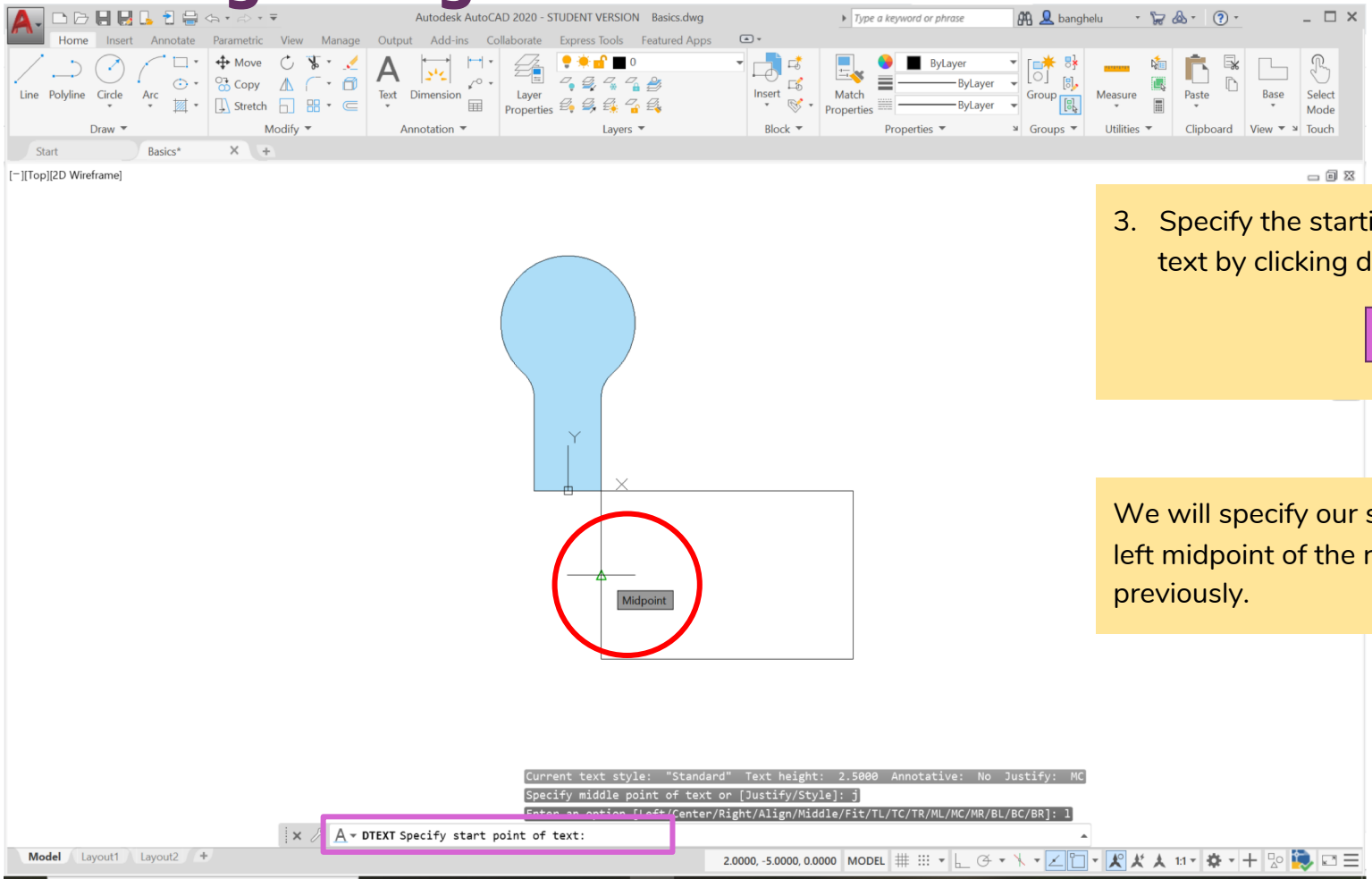


1. Type “DTEXT” into the command line and press enter
2. Type “J” for justify, follow the prompt and type in how you want the text justified (middle right, middle centre, etc.)

Next Slide

We will left justify our text:

Writing a Single Line of Text

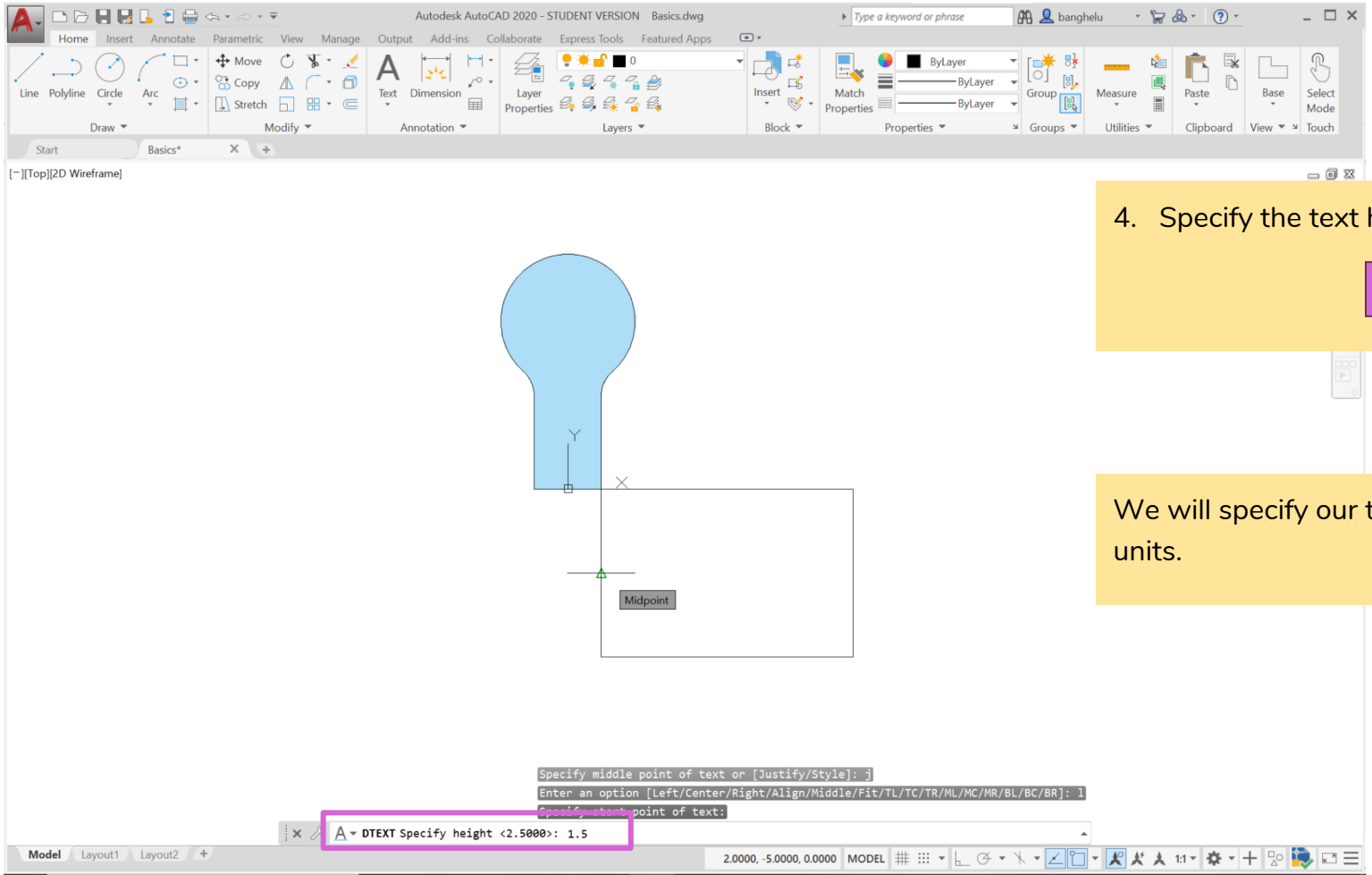


3. Specify the starting point of your text by clicking down on a point

Next Slide

We will specify our starting point as the left midpoint of the rectangle we drew previously.

Writing a Single Line of Text

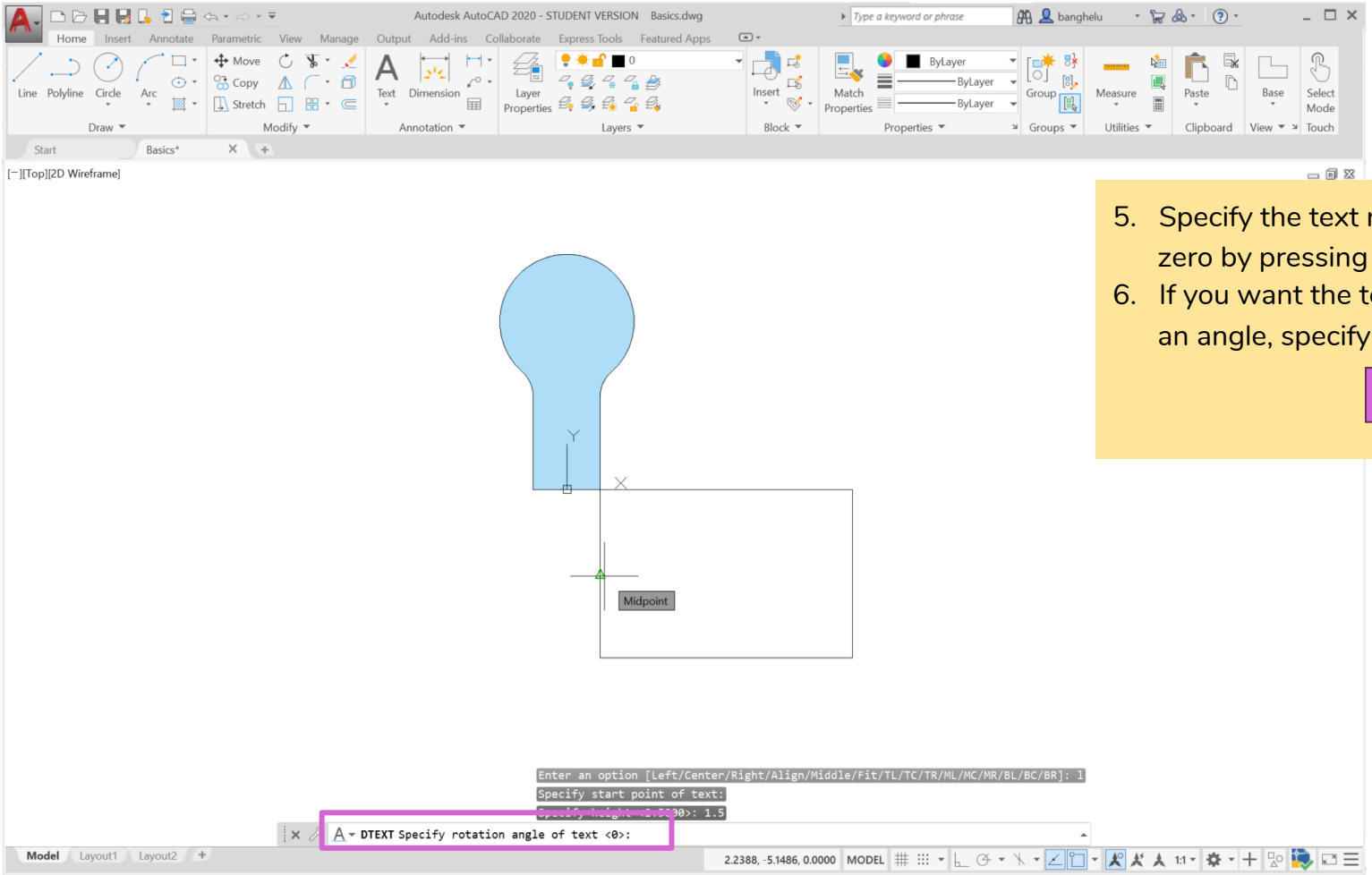


4. Specify the text height

Next Slide

We will specify our text height as 1.5 units.

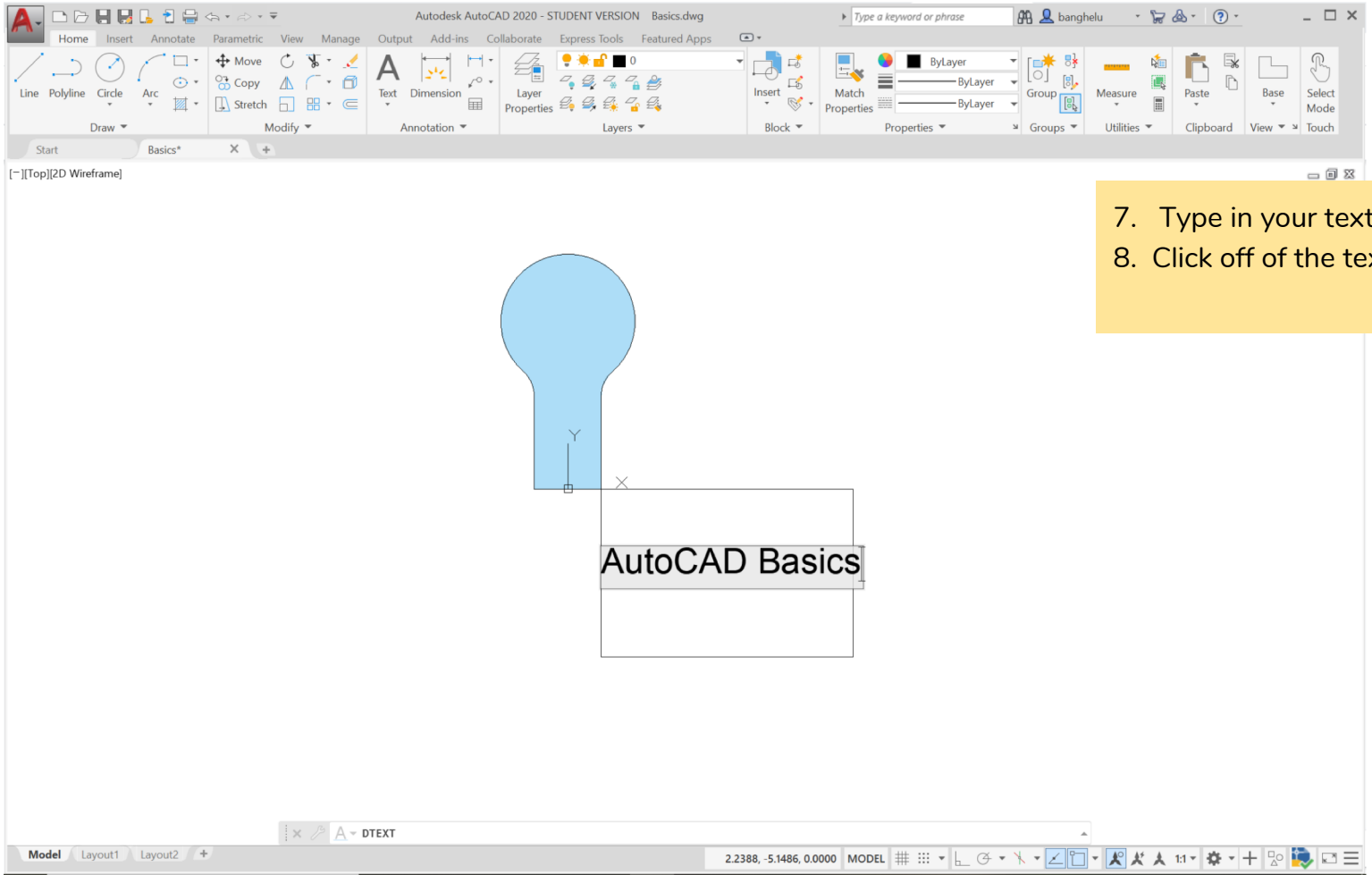
Writing a Single Line of Text



5. Specify the text rotation angle as zero by pressing enter
6. If you want the text to be written on an angle, specify that angle

Next Slide

Writing a Single Line of Text

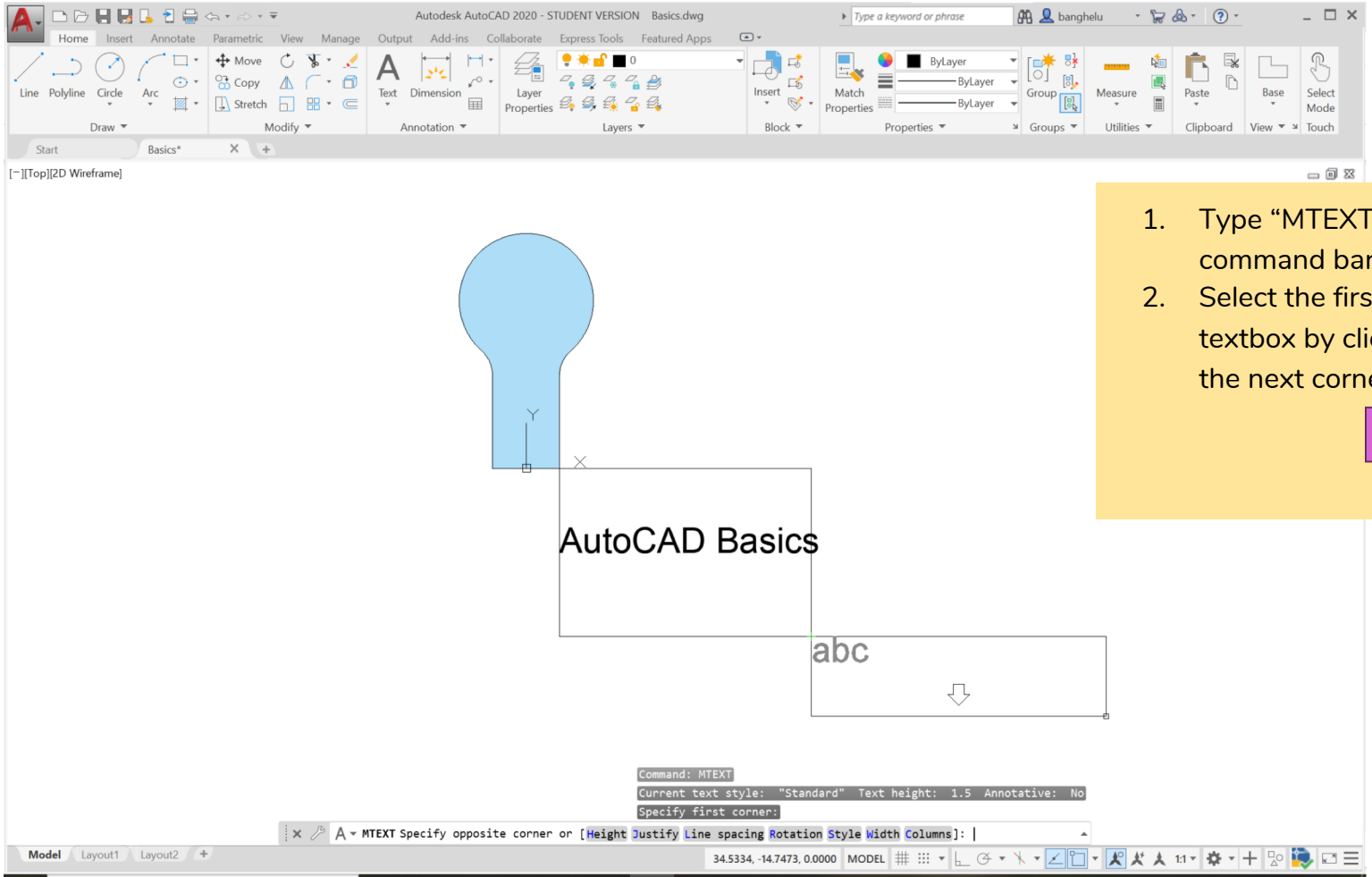


7. Type in your text!
8. Click off of the text to stop typing

MTEXT

- Write a multiples lines of text
- Decide on the text justification (if the text is aligned on the right side, left side, or center) and height
- MTEXT stands for Multiline Text

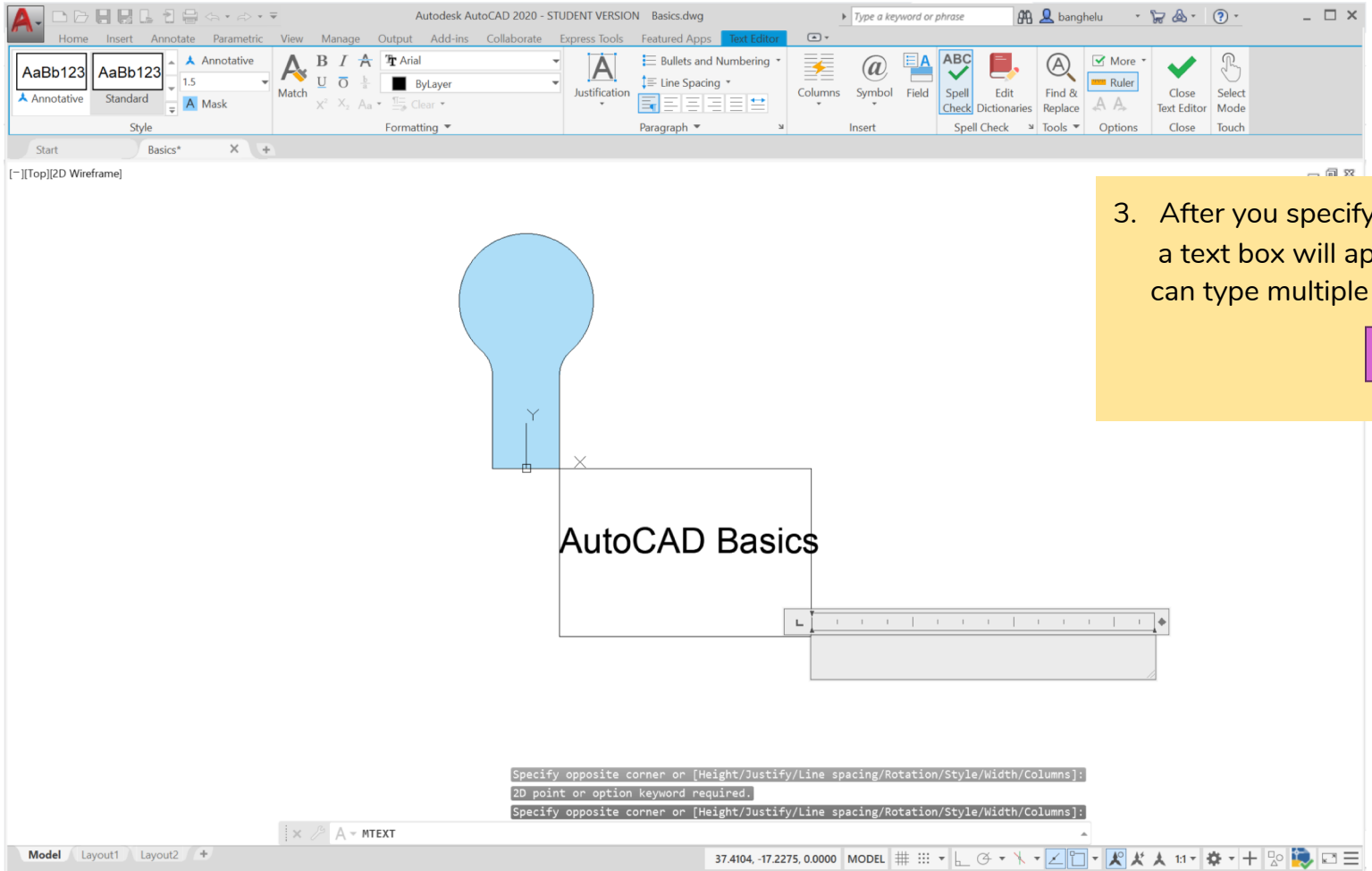
Writing a Multiple Lines of Text



1. Type “MTEXT” into the command bar and press enter.
2. Select the first corner of your textbox by clicking, then select the next corner.

Next Slide

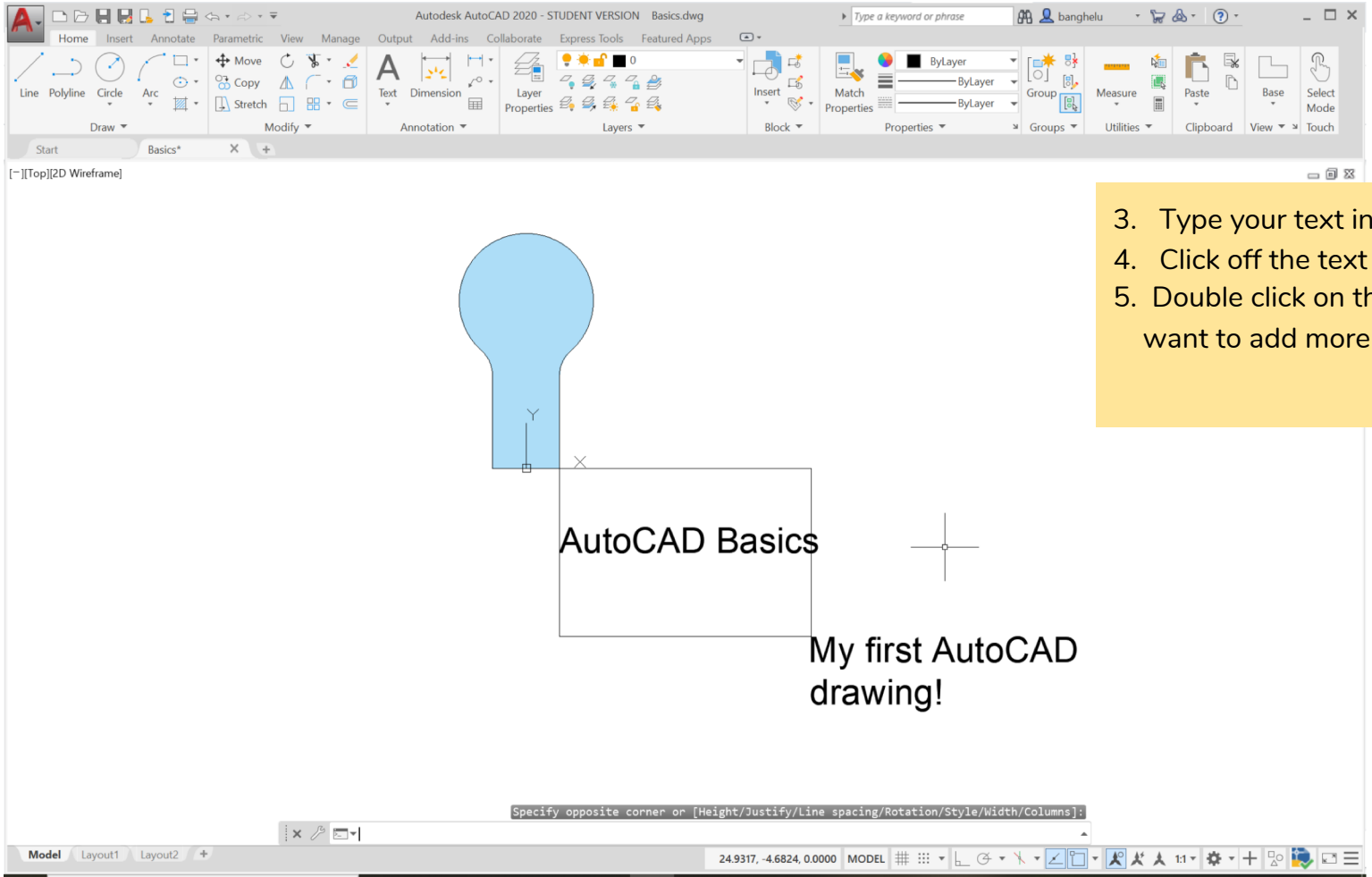
Writing a Multiple Lines of Text



3. After you specify the second corner, a text box will appear where you can type multiple lines of text

Next Slide

Writing a Multiple Lines of Text



3. Type your text into the text box
4. Click off the text box to stop typing
5. Double click on the text box if you want to add more text of make edits

AutoCAD Basic Commands

- Now that we know how to use basic commands to draw in AutoCAD, we can start making more complicated drawings!
- Check out the other AutoCAD tutorials based on your grade
- Feel free to try tutorials from other grades than your own
- Don’t forget to check out the “AutoCAD Commands Cheat-Sheet” for a list of other commands not mentioned in this tutorial!