DAVID COHN: So again, for those who came in a little late, I'm going to repeat what I said earlier. I've never understood why they have me speak on Thursday morning and do pretty much a couple of beginner classes. But this is a pretty basic dimensioning class this morning. So if you're looking for something at a more intermediate or advanced level, you want to look in the catalog and find another class to go to.

OK?

AUDIENCE: [INAUDIBLE] offended, right?

DAVID COHN: I will not be offended, no. I'd be more offended if you were here and were disappointed, because we put a lot of time into doing these. But this is a pretty ba-- it's dimension. It's the DIM command. OK. I'm not venturing very far away from the DIM command, other than a few system variables.

Be sexy? This is it. I'm getting old. This is it.

The other thing I'll admit up front-- I used to sit behind the computer and try to fake people out and make them think that I was actually doing something. I work for a company that does training videos. That's what I do all day long.

I will jump over and show some stuff in live AutoCAD this morning. But all of this is prerecorded. It's all embedded in my PowerPoint.

Please, don't give me low marks for doing that. I do that for a reason. This may be a beginner class, but I'm going to cover a lot of material for the next hour. And I want to make sure that I get through it all. So by having everything embedded in there, I can focus on what I'm trying to show you. I can use the tools that I use on an everyday basis to zoom in and show you exactly where I'm clicking and picking to make it easier for you to see it.

Also, the handout is already available online in the app. So hopefully, you've grabbed the handout. The PowerPoint presentation that I'm going to be showing is already available online in the app. And any of the drawing files that I used to record those videos are already in a zip file online.

So when you get back-- I saw that there were some instructors in the room. I used to teach at

a local community college. And I also got called in a couple of times to teach classes to teach other teachers how to teach AutoCAD. So I want to make sure that all the materials that I prepare to be able to teach this class are available to use. You can take them home and show them to the people that you work with.

So I think we are ready to start. So with those caveats over, this is what we're covering this morning. OK, so we've been doing dimensioning for years-- one of the first commands. I started working with AutoCAD version 1.4. Dimensioning came in that release. Earlier releases-- the first three releases didn't have dimensioning, so very important tool. But it's been around a long time.

But last year, Autodesk changed it. So we had dimensioning tools for years. But last year, they changed the dimensioning command. So here are the four bullet points that were available online so that you'd know what we were going to cover in this class. And hopefully, we get through everything.

So just real quickly, like I said, this is a beginner class. These are the basic types of dimensions. You can create linear dimensions that are either horizontal, vertical, or aligned with something; radial dimensions, including radius diameter and jogged; angular, ordinate, and arc length. They all are created now using the exact same command.

Now there were commands in AutoCAD before. How many of you are still using those commands? Stop using them. There's a reason why last year in 2016, Autodesk went back to a single DIM command. If you use these old commands, you don't get any of the advantages of the new features in AutoCAD.

Now you want to use the single DIM command. They gave it a nice, big button on the ribbon bar. When you use that command, you get some immediate advantages.

Here's the way you're going to dimension now in AutoCAD. You're going to start that single DIM tool. And now it doesn't matter what kind of dimension I create. I want a linear dimension, click on the object. I want it aligned, click on the object. I want a radial dimension or a diameter dimension, click on the object.

I never start another command. I stay in that same command and very quickly go through and dimension everything in my drawing. Most of the time, I never even have to right-click to pick something. Even that angular dimension, I just clicked on two lines, and it created an angular

dimension, because it saw, hey, you clicked on one thing that wants to create a linear dimension. You clicked on something else-- you must mean you want to create an angular dimension. And so it immediately went into the Angular tool.

There are a few times when you may have to right-click or press the Down arrow to select an option, because if your dimensioning a circle, AutoCAD is going to assume you want to create a diameter dimension, whereas if you're dimensioning an arc, it's going to assume that you want to create a radius dimension.

But maybe you want to put a radius on a circle. Maybe you want to put a diameter on the arc. So you might have to tell it, this is what you actually want to do. But 90% of the time it's going to assume, by whatever you're hovering over, what kind of dimension you want to create and do it for you automatically.

The other nice thing now is AutoCAD will put the dimension on the dimension layer. Whatever layer you tell it you want your dimensions to go onto, if you use the new dimensioning tool, AutoCAD will automatically put the dimension on that layer, regardless of what your current layer is. So you don't have to manage layers.

It's like the Hatch command. Right? We've had that tool in the Hatch command now for five or six releases where you can assign a hatch layer, and it doesn't matter what layer is current. AutoCAD will put the hatch on the proper layer. Again, real basic stuff.

When you create a dimension, by default, AutoCAD creates associative dimensions. That means the dimension behaves as a single object. The dimension line, the dimension arrow, the dimension text is all a single object. You don't want to ever explode it.

And by creating it that way, the dimension will react to whatever you do. So dimensions in AutoCAD, unlike dimensions in Inventor, are associative. And they're annotative.

And what that means is when you change the object, the dimension will update. So the dimension is all one object. You select it. You get grips all over it-- at the ends of the extension lines, the dimension lines, the text.

But if you change the size of the object by grip editing, the dimension updates, whereas in Inventor, when you change a dimension, the object updates. So in AutoCAD, the dimension is driven, rather than driving.

The other nice thing is that, by default, when you are snapping to things in AutoCAD, AutoCAD is going to find the object you're snapping to, not the dimension. There's a check box now.

See if I can make my laser pointer work. You've got a check box in the Options dialog that, by default, says, ignore dimension lines and ignore extension lines when you're snapping to stuff so that when you're using Object Snap in AutoCAD, you don't accidentally grab the extension line rather than the object you're trying to dimension to when you're working in close quarters.

So the new DIM tool, as I said, it was added last year in 2016. And as Autodesk likes to do when they add a new tool, they make it very prominent so you'll start using it. So the Dimension tool is nice and big on both the Home ribbon and the Annotate ribbon. And it's also got a nice little sunburst icon on it.

By the way, anybody not on subscription now? OK, so a couple. If you're on subscription, you're getting your updates to AutoCAD. So AutoCAD is now up to 2017.1. And I don't know if you've noticed it yet, but as new features start rolling out, there's going to be a little red dot next to anything that shows up in the ribbon that tells you that this is a new command. You can turn that on and off. But again, Autodesk is doing what they can to direct your attention to things in the ribbon when they start making changes or adding new features.

So again, just to reiterate, we've got those old existing dimension commands-- DIM Angular, DIM Linear, DIM Radius, DIM Diameter. They're still there. They're available in a pull-out. But don't use them, because if you do dimensioning the old-fashioned way, you won't get any of the advantages that I'm showing you this morning.

A couple of things also to remember when you create your first dimension in AutoCAD-- as long as it's an associative dimension, AutoCAD is going to create a layer called Defpoints. And it places some control points or invisible nodes on that layer.

You can't delete that layer. You can't rename that layer. Anything you draw on that layer will not print. But don't use that layer for anything else. Just ignore it. But if you started a drawing and it's only got a zero layer, as soon as you put a dimension in it, you're going to see it's got a Defpoints layer.

So let's talk about the dimension layer. Dimensions that you create using the new DIM command will allow you to specify the layer on which the dimensions will be created. You've got a choice, just like you do with hatching.

You can set it up so that it will place dimensions on the current layer. Or you can set it up to place the dimensions on any layer that you specify. But when the new DIM command is active, in the middle of the command, you can create a new dimensioning layer on the fly. So if you haven't set your drawing up with using a template where you've always got one layer that you're going to use for dimensioning-- and that's really the way you should do it-- but if you haven't done that, right in the middle of using the dimension command, you can tell AutoCAD, you know what, put the dimensions on this layer.

If you notice, there's an option right in the middle of the command. And you can just type an L, and AutoCAD will say, fine, I see that you want to tell me what layer the dimensions are going to go onto. And then you can either pick something that is already drawn on the layer you want to use for dimensions, and AutoCAD will say, fine, I'll put it on that layer, or you can type a layer name.

And if the layer name you type isn't already in your drawing, guess what? It's going to create that layer for you on the fly. And that layer will take on whatever properties exist for the layer that had been current.

So if the layer that had been current was green, the new layer that you create on the fly will be green. If the layer had a fat line type, that new layer that you create will have a fat line type.

So I'll start the layering command. Before I do-- so I've got three layers existing in this drawing already. I'll start the DIM tool. All right.

Now I'm telling it, go ahead and draw on the dimension layer. But my zero layer is still the current layer. And now I'll start dimensioning. So that dimension was just placed on my dimension layer.

I can change the dimension layer on the fly. I haven't stopped the command. I'm still in the DIM command. And now I'll go ahead and specify a new layer.

And I'll pick the layer that I had been using before. And so my next dimension will be placed on that green dimension layer. But if I use that tool again and I type a new layer name-- and what do I call it? My Dimension?-- the next dimension I create will be placed on that layer. But it'll use the set of the properties of the current layer, which, in this case, was white with a wide line type. And then I can always go back and change that later.

And notice that there's my Defpoints layer that got created now, as soon as I created that dimension. But then I can go in, and I can change the properties of that new layer that got created.

OK, so let's talk about creating dimensions. I can create any of these types of dimensions with that single DIM command-- so again, linear, angular, radial, arc length, and ordinate-- and the command will keep on going until I stop it. So I can create any dimensions I need to.

Typically, the way I work now, I will draw everything I need to draw, in terms of my geometry, and then I like working in paper space. So I'll create my geometry in model space. Then I'll go over to paper space. And I use what Autodesk used to refer to as trans spatial dimensioning.

I put all my annotations in paper space-- all my notes, all my dimensions. It works really well now. There were issues years ago in earlier releases. And a lot of people kept on working in model space.

I love to create my stuff in model space. And then I do all my adaptation in paper space, because the stuff that we create in paper space should only show up on the sheet of paper. And dimensions are one of those.

If you draw your dimensions in model space, especially if you're working in 3D-- which I'll do in a class later today-- if you create your dimensions in model space working in 3D and then you create an orthographic projection, those dimensions are liable to show up on edge. And you wonder, where'd this line work come from? So I create my dimensions on my sheet of paper.

So we'll start with the most basic kind of dimension-- linear dimension. Linear dimensions are typically placed either horizontal, vertical, or aligned with an object. I had a student in a class last year-- I was using object snap tracking to get my dimensions when I wanted to create a dimension that was horizontal or vertical along that angled line. So to create that dimension, I was using object snap tracking.

And one of the students pointed out to me, he said, you don't have to do that. Press the Shift key. When you're creating a dimension along something angled like that and you press the Shift key, AutoCAD says, oh, you want to create a horizontal or vertical dimension. And then it'll go back to horizontal or vertical, depending on where you drag your cursor.

I didn't know this. So I went back from AU last year and quickly put a video up on our website and up on YouTube showing how to do that. So now that's the only way I work with this.

So again, I'll set my dimension layer. And then I'll start the dimension command. And then I'm never going to stop the dimensioning command. I want to create a horizontal dimension at the bottom. Click on the object. Place the dimension.

A vertical dimension along the left edge-- select the object, place the dimension. Another one across the top-- select the object, place the dimension.

But in this case, if I want that dimension to be horizontal, press the Shift key, and then drag up. You want an overall dimension? Same thing-- press the Shift key, place the dimension.

You never have to stop the command. So you get to work in a very fluid fashion. And then, again, because this was the first set of dimensions I placed in that drawing, notice that AutoCAD created that Defpoints layer for me.

So here's one type of dimension that mechanical engineers typically create-- a baseline dimension where you're placing a string of dimensions, each one from the same base point. The nice thing about this command is we've got a system variable called DIM Continue mode. And by default, the system variable tells AutoCAD to place any continued dimension, or any baseline dimension, on the same layer as the base dimension.

So here, AutoCAD will actually override not only the current layer-- not only the dimension layer that you specified-- but it will place all of the new baseline dimensions on the same layer as the first baseline dimension you selected. So in this case, my dimensioning layer is this alternate dimensions layer. So if I place a new dimension, it's going to be placed on that dimension layer.

But now I'm going to create a baseline dimension. So I just Down arrow, select Baseline, click on that baseline object. And I picked the wrong one, so I just say Reselect.

And notice that it's placing the baseline dimensions on the same layer as that dimension I selected as my baseline start, even though it's not the current layer and it's not the current dimension layer, because typically, if you've got different layers for different types of dimensions, you want to place those baseline or continued dimensions on the same layer as the one you started with. And this works with continued dimensions as well.

So architects-- I'm an architect by profession. We typically dimension this way. We use a continued dimension where we're placing a string of dimensions along, say, a sequence of

walls. But it's the same command and same functionality.

We've still got this DIM Continue mode. I'm still just going to start the DIM command. So again, here's my overall dimension. And it's being created on my current dimensioning layer.

But I've got this guy over here. And when I select the DIM Continue, notice that it's placing those dimensions on the same layer as the dimension I selected to continue from, because by default, that's the way AutoCAD works. If you don't want it to do that, you can change the value of the DIM Continue mode system variable.

OK. Angular dimensions-- so an angular dimension dimensions the angle between two non-parallel lines or the angle subtended by an arc or the angle formed by three selected points. And the neat thing about this DIM command is that if you're dimensioning between two non-parallel lines, when you click on the first line, AutoCAD says, oh, I see you've clicked on a line, I think you want to create a linear dimension.

But as soon as you click on a second line, it goes, oh, well, that's not what you meant. You're dimensioning the angle between those two lines. So it'll do that by default.

So to create this dimension right here, all I had to do is click on that line and click on that line. To dimension the angle subtended by the arc, this is when I need to hit the Down arrow or right-click and tell AutoCAD, no, I want to create an angular dimension, because when I click on that arc, what kind of dimension is it going to create by default? Radial, yeah.

OK. So again, the DIM command is active. So here, I place a radius and a diameter dimension. But now I want to dimension that angle.

I click on the first line, and it wants to create a linear dimension. I click on the second line, and it immediately realizes what I'm trying to do and goes immediately into the angular dimension, then back to creating a diameter. But now when I click on that one, it wants to create a radius. And so I Down arrow, select the Angular option, and measure the angle subtended by that arc. And again, if I want to measure the angle from those three points, I have to tell it, because when I click on that circle, it goes back to wanting to create a diameter dimension.

OK. How many of you know that you can create baseline and continued dimensions as angular dimensions? All right. Makes perfect sense, right? You want a dimension, in this case, I'm going to create a series of baseline dimensions that measure the angles to each of these

offsets on this-- now I forgot what this shape-- what is this, a [? camp? ?] Thank you.

So something we do all the time-- again, I'm in the middle of the dimensioning command. And it's placing dimensions on whatever my current dimension layer is. But now I'm going to do a baseline dimension. And because it saw that I selected an angular dimension, it's going to create angular baseline dimensions. And it's going to place them on the layer of the object that I selected, even though that's not my current layer and it's not my dimensioning layer.

So it works with baseline dimensions. And it works just as well with continued dimensions. So if I want to dimension the angle between all of these holes around this wheel hub, again, maybe I want to place a linear dimension to show the diameter. So it's going to place that dimension on my dimensioning layer. But now when I say, I'm going to use the Continue option, and I click on that dimension, it goes back and starts placing the dimensions on the layer that they'd been drawn on and places them as angular continued dimensions.

OK. Radial dimensions-- so radial dimensions measure the radii or centers of arcs and circles and will optionally create a center mark or center line. So whether or not it creates the center mark or center line is based on the current dimension style.

Now in a few minutes, I'll show you the new DIM Center command, because now we finally have a tool in AutoCAD that will let us create dimension center lines and center marks that make sense when we're doing mechanical drafting to be able to actually show a center line or center mark. But when you're creating a radial or diameter dimension, by default, AutoCAD will put a little tick mark in the middle.

And it is associative now. In the past, it wasn't associative. That mark that it places in there now is associative. If you do something to move the circle, that center mark will follow with it. But the size and appearance of that center mark is based on the dimension style you're using when you create your radial or diameter dimension.

Also, it's going to have a diameter symbol, typically. If it's a diameter dimension or an R, if it's a radial dimension-- again, that's being controlled by the dimension style. So you still need to go and adjust dimension styles if you want AutoCAD to do something other than what it does by default out of the box.

So when I start my Dimension tool-- and again, I'll start placing some linear dimensions on this object. When I'm ready to dimension the diameter of that circle, all I have to do is click on it.

AutoCAD automatically sees that, hey, it's a circle, probably wants to be a diameter. Hey, it's just a radius. It probably wants to be a radial dimension.

There's where it gets controlled. It's part of the dimension style. And you can choose either mark center or none.

And if you change it, it will update in the drawing, because it's part of the dimension style. And if you move the circle, it now is associative, and the mark will move along with it. OK? Make sense?

So here is one where I'm going to have to tell AutoCAD what I want, because by default, if I want to measure the arc length, AutoCAD is going to see me click on that arc, and it's going to assume that I want to create a radial dimension. So I'm going to have to either right-click or press the Down arrow to bring up my little drop-down and tell AutoCAD, no, no, in this case, I actually want to create an arc length dimension. And again, by default, it's going to create this little hat symbol or cap symbol. That's controlled by the dimension style.

And the one thing you'll notice that after you create an arc length dimension, the next time you go to place a radial dimension, AutoCAD may assume that you want to create another arc length dimension. So this is one of those instances where once you told it you want to do one thing, you may have to tell it, OK, now I'm back to creating radial dimensions.

Here's another one-- a jogged dimension. So we'll use jogged dimensions when you're dimensioning something where the center point of the arc, you've created an arc that's so large-- maybe you're doing civil. And you're creating something to show the center line of a street that's going around a curve. And the actual center point is off the sheet somewhere.

So in this case, you'll create a jogged radial dimension where you get to control where AutoCAD is placing all the various pieces of the dimension. And if this is what you want to do, you've got to tell it, because by default, it's going to create a regular radial dimension. So you need to say, no, no, no, I want to create a jogged radius dimension.

So it wants to create a radius dimension. But I can just Down arrow, tell it, no, I want to create a jogged dimension. And then I've got several picks. I pick the object I want to dimension.

I show AutoCAD where I want to place that center point. And then I can place the dimension and control where the jog is-- so four different picks. But the next time I create a radius dimension, it's probably going to assume that that's the type of dimension I'm going to create.

OK, how many people create ordinate dimensions? A few. So you all know that when you create your ordinate dimension, the first thing you have to do is, what? Move the origin, because ordinate dimensions are used a lot in mechanical drafting when you're dimensioning a fairly complex part with lots of little cuts and offsets and holes. Rather than creating a lot of dimension strings, it's simply telling us-- my pointer not working?

So here I move my ordinate-- my 00, my UCS icon-- I'm going to move it to the lower left corner of the part. And then when I create ordinate dimensions, I'm simply specify and simply labeling the Y dimension from that ordinate point or the X dimension from that ordinate point. But it's the same DIM command.

So the first thing I have to do is move my UCS. I just right-click on it and say, OK, move it up here. And then I'll start the DIM command. So I can place some diameter dimensions on the object.

But then I'll say, OK, now I want to create a bunch of ordinate dimensions. And all I have to do is click, click, click, and position. And notice that it will automatically create jogs. So if I start dimensioning in a fairly tight location, like around that notch, I can move my cursor over and offset that dimension.

And you notice I'm not being very careful about getting everything nicely aligned, because the dimension command's also got a nice feature that, after the fact, will let me align everything and space everything. So I don't have to necessarily be real particular when I'm placing the dimensions. I can work very quickly and place the dimensions I need. And then while the command is still active, I can clean it up.

OK, so now let's talk about something that got added in 2017-- center marks and center lines. So how many of you have used the new center mark and center line features in AutoCAD? So like two or three.

This is cool stuff, because we wanted to do this for years and it was never available. So this is not the DIM command. This is the CENTERMARK command, new in 2017.

So instead of using the dimension style to add the center mark at the center of the circle or arc, if I'm actually placing center marks on a lot of holes-- on a plate, for example-- now I'm going to use my CENTERMARK command. The center marks are associative to the object that

you place them on and will size themselves to fit the object on which you're placing them.

And this is different than what you create with dimensions. And you can control the spacing of these various lines. So there's all sorts of system variables associated with this. There's like six or seven new system variables associated with these new center marks.

So there's a center cross system variable that will control just the size of that little cross in the center. There's a center cross gap system variable that's going to control just that little gap between the cross and the dimension line. And then there is a CENTEREXE that controls how far that cross or that center mark is going to extend out beyond the object. And after you create it, there's a bunch of grips that you can then use to grip edit those to make your drawings look the way you want them to look.

OK, so I've only got these two layers right now. I'll create a new layer on which I'm going to place that center mark. I'm just going to call it My Center. So this works a lot like the DIM command. I can specify a layer.

There is my new Center Mark tool. Again, Autodesk gave it a nice, big button, because it's a new tool. And single click, I get a center mark that now extends out to where, by default, it's like 0.01 beyond it. But it's a new center mark object.

If I change the diameter of that circle, the center mark moved with it. If I move the object, the center mark moves with it. Right And I've got a bunch of constraints set up in this drawing, so the other object is changing as well.

Now if you accidentally do what I just did-- I grabbed the center of that center mark and move it-- I've just disassociated it from the object. But there is a command that will let me reassociate it. And in this case, I reassociated it to the inner circle. But if I use that command again, I can easily reassociate it with the outer circle.

And then there's a second command for creating a center line. So we use a center line to show where the point of symmetry is when we're working on objects. So here's my new center line.

And you'll see in a minute when I run the video, when you start this command, AutoCAD prompts you to select two objects. And if I click there and there, it will create a center line that bisects the angle between those two objects and extends for the length of that object plus whatever the CENTEREXE.

So we've got some fewer system variables, because there's a simpler object in the center mark. But I've got a line type that tells-- and by default, it's the center line two line type. So if that line type is not currently loaded in your drawing, AutoCAD will automatically load that line type. I should have pointed that out for the center mark.

Same thing-- there is a new center line type system variable. And if your prototype drawing does not include the center two line type, it will automatically go out and find it and load it into your drawing and use it for creating the center lines and center marks.

And then you've got, in this case, the only system variable that really applies, other than the center L type, is the CENTEREXE, which controls how far that center line overshoots beyond the object that you selected. So again, if I click there and there, this portion is the calculated length. And then this is the overshoot.

Excuse me?

AUDIENCE: Is that a model space or paper space?

DAVID COHN: The question was, is this in model space or paper space? In this case, I'm doing it in model space. But I like to dimension in paper space. Now this is a situation where the center line or center mark, I might create in model space. But it'll work just as well in paper space.

OK. So again, there's my new Center Line tool. And AutoCAD prompts me for the first line and prompts me for the second line and automatically creates that center line. And that center line is associative. It is a new object. It's associative to the drawing. I can grip edit it.

If I do something to my drawing and alter the object, it will remain associative to that object. But again, it is possible to break it. If you grab that center grip and move it, you disassociate the center line from those objects. But there is a command that lets you reassociate it back if you accidentally disassociate it.

Well, you could just hit Undo, but then you'd undo-- yeah, if you caught it right away, you can undo it. And if you undo whatever you just did, you would establish the link again.

But sometimes, you may not catch it until a couple of steps down the line. You've already done something to alter your drawing. So you don't want to undo all of the other good things you did to your drawing. So you just reassociate the center line back to the objects that it should be

associated with. Good question, though.

OK. And then there are a bunch of things that we can do to modify those center marks and center lines. So I can use grip editing. I can use the Properties palette.

So center lines have five grips. Center marks-- obviously, because you're dealing now with two crossing lines-- have a lot more grips. So let's look at the center lines first. So again, I've got this CENTERREASSOCIATE command. And then I've got these two system variables.

I've got CENTEREXE, which controls the overshoot. And also, if you do use grip editing and you do make all sorts of changes and you realize that's not where you wanted to go, there's a CENTERRESET command that will set that center line back to whatever was established, based on your system variables.

When you select-- these are the grips. You get these five grips. So there's that center grip. Again, if you grab that one and move it, you're going to disassociate the center line from the objects.

But you can grab the square grip at either end, which controls the grip length. But the real one you want to grab is the triangular one. If you grab this one, you disassociate that end point. So when you use the CENTERRESET, it will not set it back to that point.

But if you grab the triangular grip and pull that out, you're changing CENTEREXE. But AutoCAD still remembers what the CENTEREXE value was. So if you use the CENTERRESET command, it'll put it back to whatever CENTEREXE was. And notice, you can actually see them in the Properties palette. So you can change those values in the Properties palette as well.

When you're dealing with a center mark, now you've got a bunch of grips, because you've got a square grip and a triangular grip at the end of each one. And again, that center grip, you don't want to click and drag that one, because that'll break it. But notice that you've got the ability to control the triangular grip separately at the end of each line.

And you can control the size of that cross, because that's a system variable. And you can control the size of that grip, because that's a system variable. Notice that, by default, AutoCAD is adding a little X on there. You all know what the X means? It's relative.

You're not specifying an absolute value for the gap. You're specifying a relative value. So if I

change the size of my cross, the gap will update to be relative to that. And it's based on paper space. And it's based on your annotative scale.

So if you're using annotative objects-- and I do work almost exclusively now with annotative text and dimensions, because I don't necessarily know the size that that drawing is going to get plotted at. This way, whatever I pick for my annotation scale, the relative size of that cross and that gap will be based on the scale that I eventually plot my drawing, not some absolute value that I assumed when I first got started.

So if you select the center line. You've got these various grips. And AutoCAD will tell you the length. And that's my CENTEREXE value.

And if I click and drag, I can change that. So if I drag it out, I can enter a value. Let's make it 1.5. Go back to the Properties palette. And that value at the end has changed.

So I can control it. I can grip edit it and do it visually. I can grip edit it and enter values. I can go to the Properties palette and change it. Center marks-- a lot more system variables, because I can control it separately at each end.

Did you catch that one? If I hover over that square grip at the center, I get a menu that pops up. And I can change all of those extensions at once so that they're equal length, rather than having to go in and grip edit them.

Now here's a situation, though. When I grab the square grip and move the square grip, I've just made a change that I can't go back from, because the length from there to there on that square grip is not part of the dimension. I've just manually changed it. And it will hang out there in space forever.

The EXE value is still the same for all four of these. But this guy here now is different than all the others. And when I reset it-- I use that CENTERRESET-- it does not go back. So that's why I say, most of the time, you want to make your change to the extension, not the absolute length. It's hanging out there now a whole lot longer than the others.

OK. We're a little more than halfway through, still a lot more to cover. Let's talk about adding and modifying dimension text. So this is a little different than the DIM command.

You all know that the dimension text gets created automatically, and it's a kind of Mtext. But if you need to add something else to that dimension, there's where the actual dimension lives. It

kind of looks like a field. You don't ever want to change that.

But if you move your cursor ahead or behind that field, you can add other text to that string of dimension. So maybe you're dimensioning four holes. You only want to place a dimension on one hole. All the holes are the same size, so you just want to say, typical. So I can add my parentheses and TYP as a suffix to that dimension. And then you can also control what that looks like using your Mtext editor.

So when you're placing a dimension using the DIM command, once you've told AutoCAD where the dimension is going to be, in the context of your drawing, there's one more click that is possible. If you simply click the last time, it's going to place the dimension where you want it, place the dimension string where you want it, using the style that's been set up for that dimension.

But if you want to go in and make a change, you can then select one of these options-- text, Mtext, text angle. And you can change the actual text. You can change the actual dimension text. I don't recommend doing that. Or you can change the angle at which the dimension has been placed.

So I'll start my Dimension command. And let's say I want to place the dimension for the diameter of that hole. But I've got four holes on there, so it's going to be typical. All four of them are the exact same size.

So I say, four holes. I move my cursor to the beginning of the string and place the prefix, and then moved it to the end of the string and used the suffix. And I'm in the middle of the Mtext editor, so I can change the style. And then I'll place that dimension.

And the nice thing is, I also created some constraints. So all four of those holes are constrained so that they're equal. And if I change one, I change them all. But my dimension updates, but the prefix and suffix remain, because they're part of that string.

I can also modify the individual dimension components. So again, dimensions in AutoCAD are driven. They're associated to the objects that you're dimensioning. If you change the object, the dimension will update to reflect the new size of the object. This is not bi-directional.

Now you can make it bi-directional. Do you know how to make that bidirectional? Anybody? OK. You saw me use constraints. I used geometric constraints to tell AutoCAD that all four holes in that plate were the same size. That way, if I change the size of one of them, all four

changed.

You can also create dimensional constraints. Dimensional constraints drive the underlying geometry. Typically, dimensional constraints do not show up when you print or plot the drawing, because they're only driving the size of the objects.

Most people then go over into paper space and place dimensions. But you can make a dimensional constraint an annotative constraint so it can be both driving and a dimension. Then if you change the dimension, the object will update.

But 90% of the time in AutoCAD, what you're working with when you're dimensioning is an annotation. So it's telling you how big the object is. And if you want to change the size of the object-- you change the object, the dimension tells you what that change is. But it is possible to create a dimension in AutoCAD by creating a dimensional constraint and use that constraint to actually dimension the object as well.

You can modify dimension components by using grips. And that's probably the easiest way. Again, I'll place dimensions on my drawings and then grip edit them and move them around and get them to look the way I want.

There's also a bunch of tools in the Extended Dimension palette in the ribbon that let you do things like create a set of oblique dimensions or change the dimension angle or right justify, center justify, or left justify the dimension. But there's also a bunch of other tools that show up when you hover the cursor over a grip and then right-click.

So I can move the dimension and place it above the dimension line. So maybe I've got a dimension style that always places the dimension inside of a gap. But for one particular dimension, I want that dimension placed somewhere else.

I can hover the cursor over the grip at the dimension text. And then I'll get a little Extended Tool Tip, and I can pick the tool to automatically put that dimension above the dimension line. So let's look at some of those.

OK. So again, here is a situation where I've created some constraints. So if I change the size of the object, the dimension updates. But I can quickly grip edit and move the dimension.

Then depending on what grip I select, sometimes the dimension moves with the extension line. Sometimes the dimension will move independently. But I can grip edit the end of the

extension line.

And then here are all those tools. So if I want to left justify that dimension, it will left justify it. My style says, place it in a gap in the dimension line. So when I change the justification of the dimension, it still places it in that gap.

I can change the angle and then continue to left justify or right justify it. Or I can make this whole string-- maybe for some reason, I want to set an oblique angle and just offset all those dimensions somewhere else. I'll undo that one.

OK. But I want to move this independent. So let's just move the dimension all by itself. So I can do that. It's still part of the dimension. But maybe I really want to have a leader on it.

You start dimensioning in really tight quarters, that's what you want to do. You've got a little, tiny gap. Let's pull an extension line off and put the dimension somewhere else.

I like this one. This is something new. You can grab one dimension and say, no, I want to work with the entire group. And then, of course, you can also flip the arrows independently at either end, all using grip editing.

OK. So remember I said when I was working with my ordinate dimensions, I wasn't particularly careful about getting everything lined up neatly. Right? I just want to get the dimensions I need on the drawing as quick as I can. And then I'll go back, and I'll clean things up.

Well, part of the Dimension command now is an Align option. So this is back in our DIM command again-- one command to rule them all. But there is an Align option that will quickly let me align dimensions.

So I don't have to worry about creating it neatly to start. I just want to get the information on the drawing. I like to work fast.

So I've dimensioned this object. And I've just kind of dropped the dimensions I know I need on there. But now I'll use the Align option.

And I'll say, OK, that's the dimension I want to align to. That's the one I want to align with. And AutoCAD will line them up for me. There's the one I want to align with. There's the ones I want to align. AutoCAD will nicely line everything up for me.

I can do the same thing with ordinate dimensions. I'm still in the DIM command. That's the one

I'm going to want to align with. And I'll do a crossing window to select them all. And I get everything nice and cleaned up. Is that nice? I'm still in the same DIM command. I could do this while I'm working.

OK, here's another one that you can do in the middle of the DIM command. I created a bunch of dimensions, and I don't like the spacing between them anymore. So there is a Distribute option that will automatically redistribute the spacing between the dimensions to clean them up for you. And again, it's all part of the DIM command.

And this one's pretty cool. So I've placed that overall dimension. And the spacing between them is wrong. So I'll redistribute this.

I can either make them all equal-- so in this case, it's just going to even them out between the first one and the last one-- or there is a dimension spacing variable. And AutoCAD actually tells you right there on the dimension line what that value is. Again, it's controlled by the dimension style. You can override it.

The cool thing here is, how many times have you ever done that? You change the annotation scale, and now your dimensions are all messed up. But I can restart the DIM command and say, fine, just distribute them again.

Go back to whatever the value is set for that spacing. And now with that new annotation scale, I can very quickly go back and change them. And now I changed my scale back. And this will work with ordinate dimensions as well.

So if I really want to clean this up, I can say, let's just space them out equally across this entire string. So in this case, it's using the offset distance. Or if I want to space them so that they're just equally spaced between the first one and the last one, just do a crossing window across all of them. And now I've got this nice, cleaned-up drawing.

And finally, the last new feature that was part of the DIM command that was added last year is how AutoCAD deals with dimensions when you accidentally place one dimension on top of another. How many times have we done that? AutoCAD will now pop up this little tool tip and say, OK, I see you just placed one dimension on top of another dimension. How do you want to deal with it?

You can move it away, in which case, it'll take the dimension you just placed the new one on

top of and move it somewhere else. Or maybe what you really meant to do was take this one dimension that had existed and break it up into two dimensions. Or maybe you want to replace the dimension you just created a new one on top of with the new dimension.

So let's look at those. So I've already got this dimension across this gap. In this case, it's an architectural drawing. But now I want to dimension to that column. And I accidentally place my dimension out there. And AutoCAD says, fine, well, what do you want to do?

Let's move it away. So it moved my original dimension string away from where I placed the new one. Well, what if that's not what I wanted? When I place that dimension on top of it, let's break it up.

Or maybe what I really want to do is replace the original dimension. Or of course, there might be a situation where I'm going to go clean it up manually later. So I can also tell AutoCAD to simply ignore. And now it's up to me to clean it up. Cool?

And then there are a bunch of other tools in AutoCAD for when you're creating the odd dimension. So here's a situation if you're doing steel detailing, you don't want to draw the whole length of the beam. So you're going to draw the beam. And you're going to put a cut line on the beam.

Well, that dimension is no longer the real dimension, because now you're just doing something representative. So for God's sakes, you don't want a real dimension line. You also want to put a break in the dimension line. So there is this DIMBREAK, or actually, this is a DIMJOG. So we can create a jog line in the dimension string.

Or maybe I placed a whole bunch of dimensions over an object, and I don't want these dimension lines to be running over the object. So there's a DIMBREAK tool that will automatically create these gaps in the dimension line where they cross either the object you're dimensioning, or they cross each other.

And there's all sorts of manual ways to do that. But there's an automatic function. And if you just do a window or crossing selection across the whole thing, it will automatically create that gap for you.

And then how many of you have-- I'm going to jump out of this for just a second and go back over into AutoCAD. So like I said, I like to work in this mode where I've pre-canned everything, because it gives me a chance to focus in.

But how many of you have had this happen when you go into a drawing and you select something to make a change, and you get this? Can you all see it? Do you like that?

You can make that go away. You know how to make that go away? There's a system variable. And it's one of the hardest ones to remember. It's called SELECTIONANNODISPLAY Don't you love auto-complete, because now you don't have to remember the names of-- I think we're up to like 900 system variables in AutoCAD. I don't want to have to remember all that.

But there's a system variable called SELECTIONANNODISPLAY. And by default, SELECTIONANNODISPLAY is set to one. And what that does is it tells AutoCAD that when you select an annotative object, show it in every single scale that it's been set up to display in, because remember, when we're creating annotative objects-- objects that change size when you change the scale of the drawing-- objects can be displayed at different scales and in different locations, depending on the scale of the drawing.

We did that when I changed the annotation scale of this drawing. But if you don't want to see all those other representations, change SELECTIONANNODISPLAY to zero. Now when you select that, you only see the representation at the scale that you're currently working in. I don't see that extra representation.

But if I go over into my annotation scale and change the annotation scale back to one to four-- and here is where I start my DIM command again-- and I Down arrow and I say, OK, I want to distribute. And we're going to use the offset value. That's the base dimension.

There are the others. And it very nicely changed all of those for me. And then if I go back again and change my annotation scale back to one to two, I'm back to the original representation. But when I select it, I don't see all of those other scale displays.

OK. So this was a pretty basic class this morning. But I hope you got something out of it that you learned something you didn't know before, even a command as basic as DIM.

So this is how to get a hold of me. And I have been saying this for 23 years that I've been speaking at Autodesk University. And I mean this sincerely.

And like I said at the beginning, I learned something new from one of the students in the class last year when I was using object snap tracking to get my dimensions to go horizontally or vertical when AutoCAD thought I was creating an aligned dimension. If a question comes up

after you've left AU, send me an email. I welcome the questions. You guys took time out of a busy work schedule to be here this week. And I want to make sure that Autodesk University was a great experience, and you can keep on learning after you leave here.

So if you've got issues-- you run into something in AutoCAD-- I've been using AutoCAD for 33 years, started with version 1.4. If you've got a question and you can't figure it out-- try the Knowledge Network first. But if all else fails, shoot me an email.

If I know the answer, I will get back to you. If I don't know the answer, I know people at Autodesk that probably know the answer. And I've got some business cards up here if you guys want a business card.

Please, fill out your surveys. If you've all got the app, it's really easy to do. It's on your app. If you didn't like this class, please don't fill out the survey.

I have been trying to win one of the speaker awards at Autodesk University for 23 years. And Matt Murphy is my boss. And if you've ever seen Matt Murphy present, he is one of the most phenomenal presenters you will ever see. But unfortunately, that means Matt beats me every single year. One year, I would like to win.

So thank you very much for being here. I can stick around for a few minutes if you've got some questions. This room isn't being used again for at least 30 minutes. This is the last day of AU. I hope you had fun last night. Please, enjoy yourself the rest of the time you're in Vegas.

[APPLAUSE]