

DAVID COHN: --a mic on now. Can you all hear? Yep. So this class, as the case with most classes at AU now-- the entire class is being recorded. So anything that you see on-screen will be in a video that you'll be able to go on to the AU website after AU and watch.

So initially, I think you have to have been registered for AU to be able to watch the video. But then after about six months, they make it publicly available to anybody. So yeah-- all the videos will be up there.

We'll switch back over to my machine and-- I used to do this as part of my AU schtick. I was the guy that always showed the viral videos. Now viral videos are just-- they're all on YouTube. But I love this one. You got to understand-- after doing this for 23 years--

[VIDEO PLAYBACK]

- This man right here is my great-grandfather. He was the first cat herder in our family.

[MUSIC PLAYING]

[MEOW]

- Herdin' cats. Don't let anybody tell you it's easy.

[MEOWING]

- Anybody can herd cattle. Holding together 10,000 half-wild short hairs.

That's another
thing altogether.

- Being a cat
herder's
probably about
the toughest
thing I think I've
ever done.

- I got this one
this morning,
right here. And if
you look at his
face, it's just
ripped to shreds,
you know?

- You see the
movies. You
hear the stories.
It's-- I'm living a
dream. Not
everyone can do
what we do.

- I wouldn't do
nothing else.

- It ain't an easy
job. But when
you bring a herd
into town, and
you ain't lost a
one of them,
there ain't a

feeling like it in
the world.

[MEOWING]

[COWBOY MUSIC]

- Woo-hoo!

- EDS--

managing the
complexities of
e-business.

[END PLAYBACK]

DAVID COHN: A whole series of those actually ran during a Super Bowl. I don't remember when, how many years ago, that is. But yeah, EDS did a series of them.

There was another one where they assembled an airplane in the air, while it was flying. And then as the various people finished their jobs, they pulled a chute and flew off the wing and disappeared. It was a clever ad series. But I love the cat herder one.

But I used to run viral videos for, like, 10 minutes before my classes started. I stopped doing it, because I didn't have the right filters, I guess. And I'd always have one in there that offended somebody.

AUDIENCE: [LAUGHING].

DAVID COHN: So I just stopped doing them. But it was like having a comedian come in and warm the crowd up before you started. But I love that one, because by the end of AU-- and I used to travel around the country with Matt Murphy and Robert Green, doing the CAD Camps that AUGI did around the country. And by the end of the day, that was the one that my little viral real ended with, because it's kind of like, that's the way we felt by the end of the conference, and we've been herding cats, so sorry, guys, but meow.

So yeah, like I said, I've been doing this a long time. And I'd like to stop doing it. But at the same time, I like doing it. If they can find kind of a mentor role for people like me and Matt and Robert and let some other folks step up and do the presentations--

This is a fairly basic class this morning, in that I don't find the tools that we're going to be covering to be very advanced. I listed it as beginner to intermediate, because a lot of folks simply don't know the tool exists. How many of you are using the model documentation tool in AutoCAD now? Nobody. Oh, a couple. How many of you are working in 3D in AutoCAD? OK, good.

I do a lot of AutoCAD work. I've been using AutoCAD since 1983. I started in the fall of 1983 with version 1.4. That was the first version that had a dimensioning command.

So I couldn't believe-- the first three releases of AutoCAD-- do you know how many commands there were in the program? 28 commands in version 1.3. Yeah. It was amazing.

So when people ask me, how do you know so much about AutoCAD? It's like the guy that lifts the goat or lifts the cow when the cow is a calf and just keeps lifting the cow. If you keep lifting that cow and the cow gets bigger, you don't realize how much weight you're picking up.

I sit down now-- I work for a company called 4D Technologies. We've got a booth here. I don't want to do a whole advertisement. But I spend my day creating training videos. I teach AutoCAD, AutoCAD LT.

I used to do our Revit Architecture course. I've turned that over to somebody else. I just this past year did our ReCap course. I am in the middle of doing FormIt 360.

I started doing Advance Steel, did a few chapters of our Advance Steel course and then dropped it for a while, and I'll come back to it and flesh out the rest of that course. Advance Steel, in particular-- I don't know how people that have never touched a program like that come in and start learning it. There's so many levels of complexity in a program like that.

So anybody that jumps into any of these CAD programs today, especially something built on top of AutoCAD-- it's a pretty daunting task. But again, today I'm only going to be looking at the model documentation tools. I consider it to be a pretty beginner to intermediate-level topic. So if that's not what you want to do during your second to last class session at AU, I won't be offended if you get up and go find another class. But I hope this will be interesting and you'll get a lot out of it.

I'm also going to let you know I used to kind of fake it. I would sit down behind the computer. And for the first two or three demos I'd show, I'd let people believe that I was actually running the software. I'm not going to lie about that anymore.

Everything I'll show you today-- I've got AutoCAD up and running. And I can bring up any of the models that I'm going to use. But I pre-record everything.

I spend my days at 4D creating training videos, and we use a wonderful tool called Camtasia Studio. And one of the nice things about using Camtasia Studio-- it's got two components, one that you use to capture the video and then another one that you use to do all the post-production stuff. And when I say post-production-- zooming in-- you'll notice when my cursor moves, it's got this nice yellow highlight on it so you can follow the cursor. When I right-click, it's a red series of radiating circles. When I left-click, it's a series of blue radiating circles.

That's all done in post-production. But it means that I don't have to worry about getting lost in a presentation and clicking on the wrong thing and then having the demo go bad. So please don't downgrade me because I canned it. I've canned it because I want to make sure that we get through everything in the 60 minutes that they've allocated for this course.

I just got done doing a dimensioning class, and it went really well, because I knew I had 60 minutes of material to show. This class has 75 minutes of material to show in 60 minutes. So we're going to move pretty quick through some of it.

AUDIENCE: Is this going to be drawn on 2D drawings?

DAVID COHN: This class is 2D drawings, but I'm not going to draw anything.

AUDIENCE: But using 3D--

DAVID COHN: No, I'm going to work with both. And I've brought models in from SOLIDWORKS. I mean, AutoCAD is great. You can consume content now from almost any CAD program.

Most of us-- what we do is we work in 3D, if you think about it. Even if all we're doing is creating 2D drawings, we live in a 3D world. We're creating 3D objects. I started using AutoCAD in 3D real early when 3D first became available in AutoCAD.

Today I use Revit. I'm an architect by profession. So I use Revit for the stuff that I'm creating on a regular basis. I even do set design for one of the local theater companies. And all of my set designs, now-- I'm working on a set right now for a show that's going up in February. And everything's done in Revit, because a set on a stage is kind of like a building. It's just inside of another building.

So what we're going to do today is we're assuming that the 3D model has been created. But now I need to create 2D orthographic drawings, because drawings don't go away. Right now there are very few companies that go directly from a 3D part to a manufactured object. And there's nobody going from a 3D-- or I won't say nobody, because we know there's that bridge being built, where, in Amsterdam now, where they're actually producing the bridge by 3D printing it on sight. So I mean, there's a little bit of that going on. But we still create 2D drawings.

I work in 3D. Even if I'm doing a simple set piece in AutoCAD, where-- I had a set last year for a show where a flying saucer was supposed to land on the stage, and the ramp came down, and the character went off and got carried away by the flying saucer. I modeled it. It was a pretty simple little wooden construction that looked good when the lights were shining just right. But it was modeled in AutoCAD in 3D. And then I had a set construction crew that needed to build it, and I gave them 2D orthographic drawings using exactly the same tools I'm going to show you today.

So the models I'm working with are 3D models that were created in either AutoCAD or in Inventor. But I could bring such stuff from SOLIDWORKS or any other CAD program, as long as I can bring that stuff in through some kind of conversion mechanism and get it into the AutoCAD drawing. But then, to create my orthographic drawings, I'm working using the model documentation tools in AutoCAD.

So that's kind of what this slide is talking about, right? In the past if we did this, we would have mentally flattened-- I mean, think about-- how many mechanical drafters, mechanical engineers, architects-- what other professions in the room?

AUDIENCE: Structural.

AUDIENCE: Electrical.

DAVID COHN: Electrical, structural. So a lot of what we're doing is 2D documentation, right? But when you're creating a 3D part when we're working on the drafting board, we would have to mentally flatten it to create our orthographic drawings. AutoCAD'll do that for us automatically.

So I can take a 3D part and create a 2D drawing or an isometric view. But I don't have to draw anything. If I built the thing in 3D, I'm going to let AutoCAD do all the heavy lifting. If you're using Revit or Inventor or a program like that, it does it naturally. Well, AutoCAD does it naturally as well if you know how to use this tool.

Everything that we do is going to start from a 3D model. And then we're going to place a base view into a paper space layout and then create projected views from that base view. That's basically the way this tool works.

So a base view is simply the first view that you place in the 2D paper space drawing that's based on the 3D model. It's created from a 3D model. AutoCAD flattens it and puts it into a 2D drawing. And then you project all of your other-- so maybe the first view is going to be a top view.

And then I'll project my side view, my front view, my isometrics. I'll place some cut lines and create section views. I'll draw a circle callout and create a detail view. It's all based on the 3D model. But it's being created in a 2D model space view.

The nice thing is over in model space, I've still got my 3D drawing. I've still got my 3D model. And if the 3D model changes, all of those drawing view are going to change. And if you build this all correctly, once you've placed annotations and dimensions, they will update automatically as well, over in Paperspace most of the time. I say 90% of the time. I'll show you a few instances where it breaks. But then there are some tools that you can use to fix it when it does break.

So if you're working in ModelSpace, when you first place a base view, AutoCAD's going to prompt you. Where do you want to place the base view? And it's going to put you over into whatever Paperspace layout you specify.

So that base view-- you don't have to worry about managing layers when you use this tool. AutoCAD's going to create the layers for you automatically. You don't have to worry about the scale. If the scale doesn't look right when you put it in Paperspace, you just very quickly tell it, that scale doesn't work. Use the scale.

If you're going to place the base view and you don't like what view of AutoCAD is using for the initial base view-- it's picking a top view and you want a front view, you change it on the fly. And then from that point on, you just work forward and you'd work the way you would logically think about creating 2D orthographic drawings.

All of the tools for creating the base view show up right on the Home ribbon over on the far right side. You'll see one button that says Base. And it's a split button, which means if you click it, you'll get two options.

One option is just create a base view from something that's in ModelSpace. The other is create a

base view from an Inventor model, and it'll open up a dialog box. Let's save the questions till the end, because I got a lot to cover, but we'll have time at the end for questions.

If you pick from Inventor, then it'll open up a dialog box and say, OK, fine, what Inventor file do you want to import? If you use the tool in a blank AutoCAD drawing-- in other words, you start a brand new drawing-- there's no content in it-- and you say, create a base view, then it'll prompt you to go find a file to bring in and use as your 3D model, because it doesn't make sense to create a base view if there's not any content in ModelSpace yet.

Come on. OK, so here is a very simple little widget in 3D. And I'm going to use that to create my first base view. So it says, from what. If I plus the press Enter, it selects everything in ModelSpace.

And then I can specify the name of the Paperspace view, and it'll create that Paperspace layout for me. I place the base view. And now I'll simply project a top view, a side view, and depending on where I move my cursor, an isometric view.

And once I've got my views placed, I press Enter, and AutoCAD creates those three views. And it really was that fast. And notice-- it's a new drawing object called a Drawing View. It's not geometry that you can modify. It's a special object.

If you need to modify the geometry, you go back to ModelSpace, right? Your object exists in ModelSpace. You modify the object in ModelSpace, all of those base views will update, OK?

Well, what happens if you want to base it on an Inventor model or you're working in a blank drawing? Blank drawing-- it says, OK, fine, go find the model. So we'll go find the 3D model that we want to bring in from Inventor. It quickly imports it. And now I'm placing my base view of that model.

But it still works exactly the same way. You place the base view, and then you can quickly create the other views. And when you press Enter the final time, it generates the view using whatever settings you told it to use. But it's a drawing view object. It's not editable geometry in AutoCAD.

What if you've created a part in AutoCAD that actually consists of several different objects? So a few years ago, Autodesk had me do a study where we were trying to prove that you could actually model faster in Inventor than in AutoCAD, or draw faster in Inventor than in AutoCAD. So I created this whole table vice in AutoCAD and in Inventor.

So I've got this nice little table vice model, and it consists of about nine different parts. I've got a base. I've got these plates. I've got the threaded access. I've got a handle. I've got these little caps. I've got these plates that go on here.

OK, so here's this model in AutoCAD. So there's nine different parts. So now I want to create a series of sheets.

And I've actually preset my sheets. I've got sheet 1, sheet 2, sheet 3. I'll select that object and create a base view of that object. Well, actually let's do the whole thing. We'll put that model in sheet 1.

But I don't want to create a 2D view. I want to create a 3D isometric. I actually want it to be shaded. I wanted it a bigger scale. So before I even place that in the model, I'll change those settings. And then when I'm all done, I've created a nice little isometric, and I can put balloon callouts on it, call out the various parts.

But now let's create other sheets that show just those individual parts. So I'm selecting just the handle. And I'm putting it on sheet 2. And it brings it in. It says, OK, I can fit it at that scale.

But let's change this now to a front view. And then I'll project a side view. And there are my parts- - and I can fit the cap drawing on there as well. So I say, let's place another base view, and it wants to bring the whole model in. Well, I don't want that.

So I'll go back to ModelSpace, clear my selection, and de-select everything except for the cap on that one end. Then I'll bring the cap in. The scale's too big, so I'll change the orientation and change the scale.

And I'll place a a front view of the cap and project a side view of the cap. And I've just created drawings of those two parts. And I could go through the process then and place each part on its own sheet and then start dimensioning all those parts. You don't have to place your base view and all the projected views at once. You could just place the base view on the sheet and then go on to do something else and come back and do your projected views later, because those projected views are simply other views projected from some other view that you've already placed.

So once I've placed any drawing view on a sheet, I can use that drawing view as the base view from which I will then project other views. Typically, you'll project all of the views from the same base view, but you don't have to. And if you project a view from a section view, you can create a

cutaway section view, which is kind of neat.

So anyway, here are two views that I've already placed. And I'll place a new projected view. This was my original base view, but I can use that as a new base view. There's nothing preventing me from doing that. But typically, I'll project new views from the same base view I initially brought in.

The projected views maintain a relationship, and there are certain controls inside the program that you can change. So you can change the hidden line style, the scale, the edge visibility, the view location. But you can't change the geometry.

Again, the drawing views are a new AutoCAD object. There is no editable geometry in the drawing view. The editable geometry is the 3D geometry that lives over in ModelSpace.

You can edit the drawing views. But you're only editing the appearance of the view, not the geometry. But you can edit it by selecting the object, right clicking, and choosing Edit View.

You can double click on it. You can click Edit in the Modify panel of the Layout ribbon and then select the view. So AutoCAD will let you start commands by picking the object first and then starting the command or starting the command first, then picking the object, or just double clicking on something or going into the Properties palette. You choose the way you want to do it.

But I can very easily select that view and say Edit the View, and my ribbon changes. And now I could change the appearance, so maybe I want it to be shaded or shaded and still show the hidden lines. Maybe I want to change the scale.

If I select the base view and move it, any views that were projected from it by default will maintain their alignment with the view, because that's just good drafting practice. But I can break that if I want to. By default, it'll let me move it left and right but not up and down, because it's tied to that view. But if I tap the spacebar or go into the spacebar, I can break that. And now it'll let me place it anywhere else. If I want to reset it, I just hover over that grip and tap the key again, and it'll lock it back into alignment.

I probably wouldn't do that with a projected orthographic view. But as you'll see in a minute, when I'm creating a bunch of section views, I don't necessarily want the section view to maintain its alignment with the section cut. I probably want to be able to place those views somewhere else on the sheet, because otherwise, the sheet could get kind of messy.

So a section view is simply a special type of projected view. If you think about what a section

view really means in drafting, you're drawing us a line that shows you're cutting through the object at this location, and you're going to reveal what's inside the object along that cut plane, and then you need to put that view somewhere else. So AutoCAD maintains this relationship, and it has certain features built into the section tool.

So when you cut a section, you're going to get a section view. That view is going to come with a section view label. You're also going to be able to get a section line and a section identifier. And if you want to create an elaborate section that bends as it goes through-- it's a job section, for example-- that cuts through the most interesting parts of the object, you can control all of that.

Constraints can be really useful. How many of you use constraints, either geometric or dimensional constraints, in AutoCAD? They can be really useful, particularly when you're creating drawing views. I typically work with the Infer Constraints tool turned on so that just by virtue of using Object Snap tracking to click on certain things within the drawing, AutoCAD will now constrain that section line, for example, to the center of the object.

But if AutoCAD doesn't get it right or I need it to be precisely located, I can go and use this tool called Symbol Sketch Mode, where I pop into this environment inside the drawing view environment. And I can now create constraints on the fly, to make sure that my section line or my detail view callout is precisely located in relation to the geometry that I'm trying to detail to make sure that the section line stays where I want it to be in relation to the object. And I'll show you that in a minute.

There are some interesting things that happen if the model that you're working with came from Inventor. When you're working from an Inventor model, Inventor knows what a fastener is, for example. And typically, when we create sections through objects, we don't want to crosshatch fasteners. AutoCAD doesn't know what a fastener is, because what you've modeled in AutoCAD is just solid objects. So it doesn't know what they are. So you can selectively adjust the objects that show up in your section view to tell AutoCAD to crosshatch it or not. Inventor would do that automatically.

And as I said, one of the cool things is once you realize that you can always go back and create more projected views, if you create a projected view from a section view, you can actually create a cutaway projected view that is a cutaway section view of your object. And because the views that AutoCAD is creating are all related to the geometry, if you make a change to the section line-- you move the section line-- your section view is going to update. If you change the scale of

the View, the section view scale is going to update. If you rotate the section line, your section view is going to update, because AutoCAD is going to maintain the relationship of all these different things that you're placing on your drawing.

And you can create literally any kind of section you want. You need a full section? Fine. A full section is just a section view that cuts through the whole object. Do you need a half section? You just draw the section line through half of the object.

An offset? Because I don't want to cut through the most interesting parts of the object. You're just changing where the cutting plane is. That's all you're doing.

When you go in and you actually start to create a section, you'll see that AutoCAD, the pull down, gives you all these choices. But those choices are just options of the command. It's still the exact same command. And you can change your mind anywhere along it.

You can also go into Paperspace and sketch some geometry, because maybe you've got weird angles and you don't know exactly what angle it's at. You can go and sketch some geometry, create some projection lines, create some construction geometry, and then use that construction geometry to create your section line. And then after creating the section line, anything you modify will cause your section to update.

And this is what I was talking about. When you use the section tool, you'll get all these options. But it's all the same command. It's just an option if you use the down arrow. If you started the command by typing and press the down arrow on your keyboard, all those options are going to be there.

So here's a little motor from a gas-powered remote control car or airplane. I want to cut a section through there. I'm using Object Snap tracking with inferred constraints turned on so that it will automatically align to the center. And as soon as I finish creating my section line- notice it's keeping me aligned, because that's good drafting practice.

And that quickly-- I created a section. And AutoCAD labeled it for me. It gave me the scale. And if I make a change to the scale, it'll update the scale. If I want to create an offset view, I can again use Object Snap and Object Snap tracking to place my section line exactly where I want it.

So I'll start out by telling it that's the view I'm going to work from, and I'll use object snap tracking to track off that center line. And then I'll pick up that center line. So I'm not clicking. I'm just Object Snap tracking. And then press Enter. And then I can grip edit that to move it where I want

it on the sheet. Pretty cool?

And then again, once I've created that, if I then use that as a base view and place an isometric off that base view, I get a cutaway section view. So there's that drawing that we just worked on. Now I'll use this view as my base view and move my cursor over and up slightly so that it creates an isometric. And I've created a cutaway section view.

Here's a situation where I don't know exactly what angle this little bracket is drawn at. So I came in first, and I sketched some geometry in Paperspace. So that's just a line drawn in Paperspace. Again, I'm working in a Paperspace layout. Now I will select-- this is the view I'm going to project from. These are the lines I'm going to use for my section line. And now, depending on where I move my cursor, it'll create a section view that follows that cut line.

Oops, let's go back. OK, I can also create sections that only slice through and look at certain aspects of my model. So the next model I'm going to work with is a shaft that's got several different diameters along the shaft, and I want to cut sections through it at different points and place a whole series of sections. So rather than let AutoCAD just-- by default, it does a full section. So it shows everything beyond the section line. If I want to just create a slice through it, there's an option to only show what the section plane is slicing through. Or I can set a distance and show only what's at the section plane and a specified distance beyond the section plane.

So here's that little hub piece. And it's a fairly interesting little model. This was fun to model in AutoCAD, by the way, because of all these little holes and things.

Here's a situation where I'm going to press the Shift key to unlock that so I can move that section somewhere else on the sheet. So that's why you would perhaps do that. And let's create another one.

Here's the notion of controlling how far you see through it. Let's create another section through that object. And I'll Object Snap track off the midpoint of that part of the hub.

Again, pull off to the left so it sees which way to project. Then press the Shift key so I can move that section somewhere else. And let's do one more. Same object off the midpoint of that part of the hub. Move to the left so it sees which direction I'm projecting. Then press the Shift key so I can move the section view somewhere else. And then once I've got them, I can grip edit my callouts and place the callout somewhere else to clean up the drawing. But that quickly, I create three sections through that fairly elaborate little hub piece.

Once you've created those section views, AutoCAD creates and adds the crosshatching to the section view using default settings. But then once those hatch patterns have been created, it's just a hatch pattern in AutoCAD. So you can go in and you can easily modify the hatch pattern using the same tools that you would normally use when you're adding hatches to an AutoCAD drawing.

So we'll just zoom in on that. And notice it actually did place the cross-hatching at some different angles. But I don't particularly like the pattern it used on those. So I can change the pattern. I can change the scale, because it's just an AutoCAD hatch pattern at that point.

When I show you the style controls over sectioning, then you'll see where AutoCAD is determining what angles to use and what scale and which hatch pattern. And it's all modifiable, because the appearance of that cross section is determined by a section view style. So just like a line type style or a text style or a style, you can control the style settings that AutoCAD uses when it generates section views and detail views. It's all part of a style that's easily configurable.

So this is what I was talking about earlier. You can use the Edit View tool. You can change the headline style, the view style, the view identifier. You can toggle the hatching on and off. You can go back and change the section depth. Maybe you forgot that you really wanted that section to only be a slice. You can change it after the fact.

So for example, let's change that one. And maybe I want that one to have a shaded background. Change the identifier. Turn the identifier on and off. Turn the hatching on and off. Change it to shaded within the lines. I can do anything I want to that view, except change the geometry.

I got rid of it, because I can actually change the style and the way it projects on that view. But I couldn't change it while I still had a projected view taken off of it, because it would have altered the projected view in a way that wouldn't have made sense, then, for that projected view. But here, I can now control the distance. So you get this dark black line that you can then either manually move back and forth or, up on your ribbon, set the actual distance that you're projecting that view beyond the section plane.

And then again, if you want to edit the view itself, if you change the section plane, the view will update. If you add a vertex to that section view-- the section line is essentially a polyline. So let's say you really wanted to create a jog section, but you started out as a full section. You can select that section line. It's a polyline. You can add vertices to that section line and turn it into a jog

section line. So you can add segments. You can flip the direction.

So if I change the direction, the view updates immediately. OK, if I change this from a jog section to a half section, now I'm only cross sectioning through part of the part. And then again, you already saw me move the identifier. You could change the identifier label. AutoCAD will simply respond to whatever changes you make to the drawing.

So maybe I don't want that to be section A. Or I want to move the section identifier somewhere else. Or I want to move it with a leader or return it back to where it was.

OK, if I change it to section 8, then it updates the label, the callout, and the section identifier. And I can change that over in the Properties palette as well. So we'll use the Properties palette to change it back to A.

OK, you can also change the section view labels. So the label is being generated by AutoCAD, but it's essentially an MText object. Your scale is on the bottom line. Maybe you want a horizontal line underneath the word "section" and then the scale below it. Maybe you want to change the size of the scale text, because right now the section identifier and the scale are both using the same text type. You can do that. It's just an MText object. So all I need to do is double-click on it.

Now, some of the information in there, you can see, is a field. So you don't want to change the text that's in there. But you can certainly change the size of the text. Or you could underline it. These are just fields inside of AutoCAD.

Now, one of the things that you probably shouldn't do is add the semicolon right there. If you want a semicolon there, it's easier to make that change in your section view style so that it's always there, because here, I'm editing each one individually. If I wanted to always include that semicolon-- or the colon rather-- I would change it as part of the style so it's always there.

Unfortunately, there's no way with the style at present to make the text two different sizes. So that one I would have to go through and manually do. And then here's that situation where if, like my table vice, it was drawn in AutoCAD and it's got some fasteners in it, when I create the section view, AutoCAD's going to crosshatch the fasteners.

If the view came from Inventor, AutoCAD would not crosshatch the fasteners because it would know that they were fasteners. But because I drew the table vise in AutoCAD, it crosshatched these fasteners. And so I need to go in and tell AutoCAD, you know what, exclude those from the

crosshatching

So I tell it which view I'm modifying. And then I'll just select the objects that I don't want to be crosshatched and tell it to exclude those. So I can fairly quickly fix my drawing so it's showing the right information. But again, if the part came in from Inventor, it would do that for you automatically.

And then here's where we control the section view style. So all the section views that AutoCAD is generating are based on the section view style. And the section of view style controls the appearance of the identifier and the arrows. It controls the appearance and line type of the section line, the section plane line, the label, the hatch patterns that are applied.

So by default, AutoCAD comes with two section view styles, one if you're working in metric, and one if you're working in imperial. So in this case, the drawing was an imperial drawing, so it's using the Imperial24 section view style. But I can go in, and I can change that or create a new one and then use that new one.

So I can essentially set my drawing styles up for my office the way I work. So I'll create a new style called CADLearning. And in that one, I'll just go in and make some changes. So over on the View tab-- that controls the section plane. This one controls the View label. Here's where I'll go in and I'll add my semicolon so it shows up all the time.

So I'm not changing the field information, but I am changing other aspects. You don't want to mess with the fields. Don't cross the streams. But I can choose which hatch pattern it uses by default and the scale of the hatch pattern. Now, maybe I want to change the color of the hatch pattern.

Now, nothing changed in the drawing yet, because all I did was create a new section view style. But now that style shows up in my menu. And if I select a view, I can assign that section view style to that view, and now everything about that view changed. So it's got the semicolon. The hatch pattern changed to magenta.

And the same thing happens with Detail View. So Detail View in AutoCAD is just a special type of view where you create either a rectangular or a circular callout from an existing view in your drawing, which becomes then the base view for that detail view. And then AutoCAD will create a detail view, which is simply an enlarged version of that view, because you've got more intricate detail going on that you need to project at a different scale so that you can label and call out the

information that's appearing in that specific spot in the drawing.

And again, the appearance of everything related to detail views is based on a detail view style. And AutoCAD comes with two of those, one for metric and one for imperial. And you can modify them or create your own.

And there are again certain aspects of that view that are part of the default. So every view is going to have a detail view boundary, which you can make visible or invisible. It's going to have a connection line, which you can turn on and off. It's going to have a detail view border, which you can turn on and off. And it's going to create your detail view labels.

So let's create a detailed view. Again, you can control the boundary type. So it can be circular, or it can be rectangular.

You can control the model edge. So maybe you want to see it, and you want it to be smooth. Or you want it to be smooth and have a border around it or smooth with a connection line. So these are all based on different drafting standards around the world. So Autodesk is giving you a choice of which drafting standard you want to use-- or jagged.

And again, I typically work with Infer Constraints turned on so that when I create a detail view callout, that call out is related to the underlying geometry. But if it doesn't get it where I want it, I can always pop over into Symbol Sketch Mode and add my own constraints manually to make sure that it is in exact relationship to the model that I've been creating.

And then once you create it, you can modify the boundary. You can modify the view labels. If you rotate the parent view, the detail view updates. So literally, any time you make a change to anything at a higher level in the hierarchy, anything below that will update to reflect the change you made.

Go back to ModelSpace and make a change to the 3D geometry-- everything in all the views you created update. Take a given view from which you then projected other views and make a change to that view-- all of the views that are based on that view will update. So it just follows things down the tree.

If you delete-- and I should have mentioned this before. If you delete a section line, what's going to happen? Section view's going to go away. If you delete the detail view callout boundary, detail view's going to go away.

And again, when you use the tool in the ribbon, you'll see two different options, circular or rectangular. They're just options for the same command, and you can switch back and forth between them interchangeably. It's the same command.

So I've got some stuff happening in the inside of the nozzle in the section view. I want to show some of this detail where all these gaskets are. So I'm going to create a detail view of this view.

I'm using Object Snap tracking so that I'm always aligned with the center axis of that view. I create that detail view callout. And now I can choose what type of model edge to use. I can choose whether or not it's got a connecting line or not. I can choose what style it's drawn in and then place it on my sheet and finish the view, and when I press Enter, the final time, AutoCAD goes ahead and generates my callouts and everything for me.

When you think about the amount of time we spend creating this stuff, if you're doing it manually and you can do it this fast by just letting AutoCAD do all the heavy lifting. And again, if I go in and I make changes to it, I can propagate those changes down through the model. So I can change the appearance of the view itself. I can change whether or not it's using jagged edges, whether or not it's got a border around it, whether or not the callout is there, whether or not it's crosshatched, because it's a detail view of a section view, so I can turn the crosshatching on and off. I can change the label. I can get rid of the connecting line.

And as you would expect, if you modify the boundary, the detail view will update. So you change the center point, the view's going to move. You change the size or the shape, it's going to update to reflect those changes.

So I select it, and it's kind of like a polyline. I've got these grips. I change it to a circular edge. Change the size-- the view updates.

And then the label is based on the view I've created. So I can move the identifier. I can change the identifier, either using grip editing onscreen or in the Properties palette. So maybe I want to take it and drag it out of the way with a leader line.

Changed my mind, I can reset it back to the default position. Select the detail view to make it active. And I can change the identifier up there in the ribbon. So we'll change it to B. Or I can pop over into the Properties palette and change it over there. We'll change it back to A. A.

OK, and changing the detail view label-- same thing we did for the section view, so I'll skip over that. Again, the appearance of the detail view is based on a detail view style. So you've got

another set of controls for detail view styles. That one only has three tabs, whereas section view had four tabs, a little less to control in here.

But you can control all these various aspects of the detail view-- what color, what text style, the position, the boundary line color, the identifier text style, identifier text type, whether or not the colon is included in there. So again, AutoCAD comes with two by default. I'm working in imperial drawing, so it's only using the imperial style. But I can go in and create my own.

So I'll create a new one. Again, I'll just name it after my company. And then I can change any aspect of it.

When I create that style, it now exists in the drawing. But it doesn't change anything that was already in the drawing unless I take one of the objects I already created and tell AutoCAD, I want this now generated using the new style.

OK, so again, I'll add my colon in there because I like to have a colon after the scale. But unfortunately, I can't make the two pieces different size. But it didn't change it yet. But now I can come up there, and my new style is available. When I change it to the new style, then it's going to go ahead and add that call. Make sense?

OK, so let's look at some really fun stuff. OK. So I kept mentioning this thing called Symbol Sketch mode. AutoCAD'll do a pretty good job when infer constraint is on of making sure to place constraints where they should be. But if it doesn't get it right, you can pop into Symbol Sketch mode place constraints manually.

The neat thing about working in Symbol Sketch mode is that you can also create construction geometry in Symbol Sketch mode. But the construction geometry is only visible when you're active in Symbol Sketch mode. When you exit from Symbol Sketch mode, the construction geometry is not visible in your drawing, and it's not going to plot.

So you can do things in Symbol Sketch mode to create what you need in terms of reference geometry and not have to worry about it somehow showing up in your drawing. So I say, OK, I want to add additional detail, and it says, hey, do you realize you're about to go into Symbol Sketch mode? Now the only view that's active is the one that I selected. And now I can say, you know what? I always want those section lines to be horizontal and vertical. I want that section line to always be located a specific distance from the edge of that object. And I'll go in and I'll specify that distance.

And now I'll say, OK, you want to save your changes? Yes. The dimension I placed on there is not visible. I placed the constraint. It's only available in Symbol Sketch mode. But now, it's controlling where that section line is placed.

And I could do the same thing with the detail view. I want that border to always be centered on the center of that hub. And I want it to always be a specific distance away.

And so I'll create all of that information in Symbol Sketch mode and then save my changes. And when I pop back out into the drawing, everything I created in Symbol Sketch mode is no longer visible in the context of the drawing that's going to get printed, but it's controlling the way things now look in my detail view. Cool?

OK, there are a few other controls. There is a dialog box that lets you set up some of the basic information that is controlling the way things look. These are your drafting standards.

For example, in the US, we use what's called third angle. So it's top view, front view, side view. In Europe, they use first angle. So they place top view, front view, side view. So AutoCAD is built to be able to accommodate both of those styles.

If your part is coming from Inventor, AutoCAD knows that if you have fasteners, the fasteners have threads. It knows to represent threads a certain way, and you can choose which way it's going to represent them. That is really only valid if what you're bringing in is an Inventor model. If you've created a fastener in AutoCAD, AutoCAD doesn't know it's a fastener. AutoCAD just draws lines, arcs, and circles. It really has no notion of what it is you've created. So that one's only valid when you're bringing in something from an Inventor model.

The shading preview is really only a preview. And the preview type is really only a preview. If you're working with a really complex model, it may help AutoCAD behave a little faster, but it's not going to change the way it ultimately looks once you create your drawing views.

OK, so everything we did was based on a 3D model that got created over in ModelSpace. but then we created these Paperspace views that are not really views of geometry. They're drawing views, where AutoCAD is generating the geometry.

What happens when something changes? Because we all know that we'll get all the way down the process. We've created all of our working drawings. And then the engineer says or the designer says, need to change the size of the part. I don't want to have to go back and recreate

those drawing views. I want AutoCAD to do the work for me.

So there is a function in AutoCAD that says if you update the model, just go ahead and update the drawing views. I'm going to turn that off. By default, it's turned on. I'm going to turn it off just for purposes of our demonstration so that you can see what is actually happening.

But typically, that is on. And so if you make a change in ModelSpace, next time you pop over to Paperspace, all your drawing views are going to update to reflect the change. But when auto update gets turned off, then AutoCAD will alert you if a change occurred and give you the opportunity to decide, did you really want to make that change, or was that a mistake? Do you really want to update all the views now, or do you have so many different drawings involved that it might take 10 minutes for AutoCAD to go through and update all the geometry, so you can choose when to let it do this.

If you turn auto update off, then if you make a change that causes a drawing view to no longer reflect what's in the model, you'll get this nice little alert balloon that tells you the model has changed. And then you have a choice. There's a link in there. If you click the link, it'll update all the drawing views. If you simply close the dialog box, though, then you'll see red marks around any views that are no longer up to date. And you can selectively tell AutoCAD, update this one, but don't update that one, because maybe you want to do some other work before you update that view.

So back to our very simple model. I'm going to turn auto update off. By default, it's always on.

Now I'll come back over to ModelSpace and make some simple changes to this model. I'm just going to use the Control key to select that face, and we'll make the part longer. But that's going to affect multiple views.

I didn't save anything. As soon as I pop back over here, AutoCAD's telling me that the model has now caused those drawing views to become outdated. Again, if I click that link, all the views update immediately.

Let's go back and see what happens if I simply close that balloon, OK? There's a little red marker down here in my tray, and if I click on it, AutoCAD shows me the commands. I've got the same commands available on the ribbon. I can update all the views, or I can update individual views. And each view that's no longer up to date has this border.

So I can update just that one view, or just those two views. Those two views now update to reflect the fact that this part got longer. These two views are still out of date. And then if I want to, I could use the same tool from down there and update the rest of them. Once they're all updated, I no longer see that little alert down in my task tray.

Well, let's go even further into this, because we're not only going to create these views. Eventually we're going to add all sorts of annotations to those views. Then what happens when the model changes?

I've not only now created all these views back over here in Paperspace. I've annotated the heck out of this thing. And now my model changes, OK? AutoCAD'll take care of that for you as well.

So we're going to do two things here. I'm going to turn off auto update so that you can see when the change occurs. And I've also got this thing called Annotation Monitor. And by default, it's always on. And if something changes in the model that causes one of your annotations to no longer accurately reflect the model, then not only is AutoCAD going to tell you that the drawing view is out of date. It's also going to tell you that the annotation is out of date.

And now again, you have the choice of letting AutoCAD fix it for you. And I don't recommend that you let it do that, because in this case, AutoCAD sees that an annotation has become disassociated, and its solution is to throw it away. I don't necessarily want to throw it away. Maybe I just want to fix it.

So I typically just close that balloon. And I'll go in and then tell AutoCAD, make this change. Don't make this change. And I'll selectively go through it. So let's look at that.

Because what it'll do is it also will-- if I simply close it, it'll put a little badge on each annotation that it sees no longer refer accurately reflects the model. And then you can go hover your cursor over that badge and selectively tell AutoCAD, fix this one by throwing it away if it doesn't make sense anymore, or let me manually fix it if it simply became disassociated.

So here is my annotated drawing. I'm going to turn auto update off so that you can see the change. And now we'll pop back over to ModelSpace. And I'll use my Control key to selectively pick that top face, and we'll move that thing up and make that little hub taller by-- what did I change it-- 15, 25.

But over here, let's get rid of the slot. So again, I'm simply using the Control key to select the face. And then I'll orbit around so I select the corresponding face on the other side of that little

slot.

And because it's a solid model, if I then delete it, AutoCAD'll heal the model back up. So I've eliminated the slot, OK? So interesting things that you can do with the solid modeling tools in AutoCAD.

But now, it's telling me, hey, the model's changed. So the first thing I'll do in this case-- I will let it auto update all the drawings. But now it's saying, OK, but annotations have broken. I don't want it to fix it automatically.

In this case, I can safely get rid of this one. The slot doesn't exist. So we'll just delete the dimension of that slot, because that one doesn't make sense anymore. But over here, because I raised that hub and something wasn't connected quite right, I'll simply reassociate the dimension. And now my dimension's accurate, and again, my drawing now reflects the true nature of that underlying 3D model.

Wow. So last year this ran 15 minutes over. We got it. We got eight minutes left. What do you think?

AUDIENCE: That's good.

AUDIENCE: Nice.

DAVID COHN: OK? So I tell everyone-- I've been doing this a long time. This is actually my 25th Autodesk University to speak at, because the year that there were four of them, I taught at two of them. This is the 24th Autodesk University I simply attended the first one, and then I said, this is cool. I want to be part of this. And then I used to do the AUGI CAD camps AUGI was doing the CAD camps that traveled around.

This is my real contact info. I learned things from you. I hope you learned things from me. And if you have a question that comes up when you're using AutoCAD and you can't figure it out, please feel free to email me. I mean that.

Last year I taught dimensioning class, and I did something, and a student said, you know, there's an easier way to do that. And they showed it to me, and I've incorporated that into the stuff I do for 4D Technologies now. So I want you to feel free to be able to contact me.

I also want you to feel free to give me all fives. If you don't want to do that, don't fill out the

survey. I joke about it.

But I work for Matt Murphy. I don't know if you've ever seen Matt Murphy's class, if you've seen him present. He's a phenomenal presenter. And Autodesk gives out awards every year for the best speaker. And I've never won because I lose to Matt. I'd love to win sometime.

I hope you enjoyed this. I hope it was informative. Please do fill out your feedback questionnaire. If you've got more questions, I think the answer bar is still open. But feel free to email me. I didn't even put a company ad in there. I'll go back to my contact info.

I've got business cards up here if any of you guys want a business card to get back in touch with me. And we're a couple of minutes before 11:30, so if you want to stay, if there are some questions. I know you had a question earlier.

AUDIENCE: Does it work with xrefs?

DAVID COHN: Does it work with xrefs? Yeah, as far as the geometry goes, yes. The geometry can come in from an xref. The only thing that gets a little crazy is if you want to place views, so you cut the section on one sheet, AutoCAD doesn't want to let you put the actual section on another sheet. Same thing with detail views. Whatever sheet you actually cut your section line or create your detail callout, you've got to create any derived view on the sheet from which the base view came from. You can't put that derived view on a different Paperspace layout. That's one of the few restrictions to working with this. But it's all AutoCAD geometry. And so yes, you can use xrefs.

AUDIENCE: I've got a quick question. So when you bring the new base from a different file, would it be worth knowing that on the sheet-- and you need to make changes-- how do you know--

DAVID COHN: OK, so the question is, if your base view came in from a different sheet-- No, the underlying geometry that you are creating drawing views from must exist in ModelSpace in the drawing in which you're working. It can be xrefed in. And then if you jump over and make a change to that xref geometry, it will propagate through the xrefed drawings. But there's no way to create drawing views from geometry that isn't somehow in ModelSpace in the drawing that you're working on.

AUDIENCE: So if you have an xref into that drawing, it could still do all your views and all that stuff?

DAVID COHN: Yes, so the question was, if you've got some geometry that you've xrefed in, can you still do this? Yes, as long as the xref is in ModelSpace, you can use everything I showed you and create your

detail views. The only way to modify the geometry that you xrefed in would either be to activate that xref in the context of the current drawing.

Remember, you can always tell AutoCAD that you want to modify the xref in the context of the current drawing. There's a tool for doing that. Or you could go back and open up the drawing that you xrefed in, modify the 3D geometry in that drawing. And as soon as you update the xref in the current drawing, anything you then derived from that xrefed geometry would still cause all of your drawing views to update.

But somehow the geometry that you're working with must exist in ModelSpace. Either it came in from Inventor. Or you imported a CATIA or a SOLIDWORKS file in there in some format that AutoCAD can convert from. Or it was created as AutoCAD geometry in some way, shape, or form living now in ModelSpace. Yes.

AUDIENCE: Is there a way that to actually convert the drawing views to actual geometry?

DAVID COHN: Is there a way to convert the drawing views to actual geometry? There is, but it would get kind of nasty. I mean, one of the ways you could do that would be to export it in some other format. You're basically then flattening it out into dumb AutoCAD geometry. But then it's not going to update, right? Right. Yeah. Yeah, conceivably you could do that. I mean, think about the tools that Autodesk has recently added with PDF. You could plot to a PDF and then import that PDF and convert it into AutoCAD geometry and go off and modify it again. Question on this side.

AUDIENCE: Inventor-- so you bring a model from Inventor. But if you update your Inventor model, does it automatically update in AutoCAD?

DAVID COHN: Interesting question. So the question was, if you brought in an Inventor model into AutoCAD and then you went back to Inventor and made changes to the Inventor model, will it automatically update in AutoCAD? I don't know if it's automatic, but Inventor has bidirectional associativity to DWG.

So in Inventor, if you have told Inventor that your output format is DWG-- I'm not quite sure why you'd do it. But if your output format from Inventor for creating drawings was DWG, your DWG's would be associative.

I don't know that you'd want to do it for only one reason. Inventor has always had these tools. This is the way Inventor works natively. You model in 3D and create two-dimensional views. Some people like to create their two-dimensional views as DWGs. And so Inventor has the ability

to do that. And within the context of Inventor, those views are bidirectionally associative.

If you make a change in the 2D view in Inventor, you can push that change back into the 3D model. When you go from Inventor to AutoCAD, you're essentially importing the Inventor geometry right into AutoCAD's ModelSpace and then creating the views. So I'm not sure-- if you update the Inventor model and then bring-- I don't know if it maintains an active link. I think it's a one-way import. So I don't think you can do it. Yeah.

AUDIENCE: So it's one way.

DAVID COHN: It's one way. Yep.

AUDIENCE: So the reason that you would want to do it, especially at my company-- we have a set standard that we use for all of our [INAUDIBLE]. And in Inventor drawing, it's really difficult to make it look right.

DAVID COHN: Yeah.

AUDIENCE: So if you do primarily everything in AutoCAD, then that's why you'd want to do modeling in Inventor. It seems so much easier to do all your modeling in Inventor, then bring it over to AutoCAD.

DAVID COHN: Right. But Inventor will let you create all of your orthographic views and save them as DWG files. And they are they do remain associated to your Inventor model. So this tool is really more for the person who doesn't have access to Inventor, who doesn't have access to SOLIDWORKS or CATIA or some other CAD program. They're consuming 3D geometry that was created by someone else. But now they've got to document it from the standpoint of the manufacturing floor, right? So that's why you've got that ability of being able to bring your Inventor models or your SOLIDWORKS models into AutoCAD.

AUDIENCE: I have a quick question.

DAVID COHN: Yeah. So speaking of standards, we use bubble callouts and stuff like that.

AUDIENCE: Is there a way of updating the way that AutoCAD does its detail callout to look differently?

DAVID COHN: So the question relates to the actual appearance of the callout, the circle or the rectangle. Right now, no. You've got two choices. They can be circular or rectangular. You can do some things with the line type and the line style when you're creating your detail style, your detail view style.

But it won't let you do anything other than a circle or the rectangle.

AUDIENCE: Yeah, can you just keep it in the Symbol Sketch mode?

DAVID COHN: Yeah, because Symbol Sketch mode-- all you're doing is Symbol Sketch mode is creating construction geometry, OK? When you exit from Symbol Sketch mode, you saw-- that geometry is no longer visible inside of what is eventually going to be printed. And that's intentional. You may need to go in and create some additional construction geometry just to get everything to stay where you want it to stay. But it's not something you ever want to show up inside of the printed output.

AUDIENCE: So if I put those little lines for the section type-- if I put that on defpoints, will that not print?

DAVID COHN: No.

AUDIENCE: OK.

DAVID COHN: Yeah, defpoints never prints. And the class I taught before this was about dimensioning. And I had an instructor that said his students are always drawing things on the defpoints layer. I said, I wish Autodesk would just make that layer unavailable to anybody and not even show up in the layer list, because it doesn't plot. But it's there. And you can make it the current layer, which means you could draw on it. You can do all sorts of things with it that it's not meant to do. Why make it exposed at all? Or create a system variable so that by default, it's hidden, and you have to know about it to turn it on.

By the way, I will give a call out to my friend JC Malitzke, who's-- I don't know if he's still teaching, but he used to teach at a community college outside of Chicago. He did a hands-on lab of what I showed you today, yesterday. So if you didn't get an opportunity to take that class, that was also recorded. So when you get back home from AU, go look for J.C Malitzke's course. His last name is M-I-L-T-K-Z-E, I believe.

But anyway, look for J.C.'s course. He did a hands-on lab, essentially on the same stuff I showed you today. And it would be another great way, if you want to get more information about what I showed you today. Any other questions? Yep.

AUDIENCE: How long has this been around?

DAVID COHN: This tool?

AUDIENCE: Yeah.

DAVID COHN: Oh god, let me think. All the AutoCAD versions kind of blur. I want to say that it's been around since at least 2009 or 2010, but it could be even older than that.

AUDIENCE: Wow.

DAVID COHN: Yeah. And it hasn't changed a whole lot. But you see from the number of people that have really never used it before. That's why they have me keep coming back and doing this class.

AUDIENCE: I've never been shown--

DAVID COHN: Yeah.

AUDIENCE: [INAUDIBLE].

DAVID COHN: Yeah. It's one of those hidden gem tools inside of AutoCAD that, once you see the way it works, it's not going to solve 100% of the issues, but I can get about 90%, 95% of what I need to do creating two-dimensional drawings. Like I said, I do a set piece. And I draw it up in 3D in AutoCAD, and then I need to put dimensions on it so that my construction crew knows what to build. I do it in AutoCAD.

I do a lot of my set design now in Revit, because a set is basically just a building built on a stage. But if I've got set pieces that they've got to build, I've had pieces that they roll it on stage on a wagon and then pieces fold out from it, and I need to show them, how do you build it and where do you pivot it and get it to all fold back up. I'll draw that as an AutoCAD thing and save it and then bring it into my Revit model so I show them how to move it around on stage. But yeah. OK? Hey, thanks so much. Enjoy the rest of AU.