NINA SHAO: So welcome to this class, and it's very exciting to have all of you together to learn Sheet Metal stuff. So firstly, we'll do some self-introduction. I am Nina. Been working as experience designer for Inventor and working for Autodesk for more than 10 years.

So I worked for several projects like modeling, assembly, tooling, BIM content, and Sheet mental. And River is our co-speaker and he's the expert for Inventor and also worked for many projects like tube and pipe and frame generator and Sheet Metal. So he contributed a lot of tips and experience for this slide and for this class.

So both of us have a background for mechanical design. So we went to Autodesk and grew up with Autodesk and learned a lot from our great customers. And you are part of them. So first thing, let me say thank you for all the continued support and greats comments.

And then let's go back to our topic-- Sheet Metal. So before we start the class, I want to ask a question. So how many have using Sheet Metal today in Inventor?

Wow, cool. OK, so for the summary for this class, that the title for this class is to learn top down design with [INAUDIBLE]. And during this class, we will comb the whole Sheet Metal system, since everything in the system will help you on the top down design. So this is the objectives for this class. Firstly, we will learn how to set up in the manage Sheet Metal rules. And the second is how to make use of Sheet Metal features.

So as we know that although our Sheet Metal has existed for many years, but some customers or some engineers still are using modeling tools to create Sheet Metal feature. So let's go through some good tips for these features and to ease your work. And the second one is go to make top down design with Sheet Metal.

And the first one that when you'd pay attention to is get tips to ease your work. So these are great tips. So if you pay attention to these tips, that you will get a chance to get a gift in the front of the class. And finally, I will tell you where to find us doing the [INAUDIBLE] time.

So most of us know that Sheet Metal works for industrial machinery and for these [INAUDIBLE] in [? closer ?] or cabinets or even transportation truck, a lot of machine stuff, Metal stuff. But also Sheet Metal can work for these paper stuff and it has the packing box. And also we can make [INAUDIBLE] stuff with the Sheet Metal.

And we'll make sample VR glasses. So we will show this model in this class in the top down design section. So that's the definition.

When do you need to use Sheet Metal for the final, single piece part for manufacturing? So pay attention to my description as [? a ?] single piece part for manufacturing. So you'll have a uniform material.

You'd have uniform thickness. And also it needs some work for bend and folded. So then if you meet these requirements, you can choose to use Sheet Metal.

But if you have any definition beyond these requirements, we can talk more offline after class. So who used the Sheet Metal from R3? Whoa, thank you so much.

So that's the highest per volume under Sheet Metal. So it has being 16 years old for Sheet Metal. And we just gave investment on Sheet Metal design for some key milestones. Like for R6, we provided a [? two ?] and for 2010, then we have a [INAUDIBLE] provided.

And most important, from 2016, we provided [? multi-building ?] Inventor. That would give opportunities for you to make the quick top down design. And in 2018, so I cannot talk too much about 2018 since [? for ?] confidential problem.

But we welcome you to join us in beta four room. And for the new functionality, [? we'll ?] addition, review, and provide comments. So we have some engineers in [INAUDIBLE] and you can find us if you want to join.

So by looking at the Sheet Metal system as a tree, it will be easy to understand how Sheet Metal works. So we call them Sheet Metal rules. It behaves like the root of the tree. It supplies the food and waters to the branches and into the leaves so everything can live. For solids, it becomes like main branches.

So from 2016, Inventor has multi-body. So who already used 2016 now? Oh, cool.

So I started using multi-body. OK, cool. So it's a good standing for multi-body support in 2016.

And in these features, we have like leaves. So these features then only need Sheet Metal features. It also means all general modeling features since it's powerful tools and that you can use them anywhere. Just follow the Sheet Metal rule.

So a little different-- a little special one is override. It override the behavior in Inventor. And it look-- we can take them as the weather. It's like the red [? necklace ?] [INAUDIBLE] or sun or wind and it will impact the growth of the tree.

By default, you always get the rules from the Sheet Metal rule. That's like the [INAUDIBLE] rule, bend creation, and corner creation. But in each feature, you have the opportunity to customize these rules separately. So that's overriding behavior.

So let's start with the root. It's Sheet Metal rules. So since most of you have the Sheet Metal experience, so we can go through this very quickly. So we will learn that how to set up Sheet Metal rules and to understand and set up and follow rules.

And primarily, we will say how to share the rules with team once you set up all this stuff. So you can access the style and standard library from this command style editor. Also, you can access this from the Sheet Metal default dialogue.

There is a little icon that can help you to access this dialog. So that's the [INAUDIBLE] in the first Sheet Metal design. And there are two main parts. The first one is the Sheet Metal rule and the second one is unfold rule.

So for Sheet Metal rules, there are three main parts-- for the Sheet plate, for the bend, and for the corner. So for the Sheet, that [? they add ?] two essential definition you need finished. One is the material, another is the thickness. So this material dropdown list that it's shared in Inventor anywhere.

So you can customize your material and access it from here. Oops. So for bend, the most important thing is to define the release shape and the bend radius and the bend transition.

So there are three types of relief types in Inventor. It's a round, [? tier, ?] and straight. So for corner, it's similar with bend that you need define the relief type and the relief parameters.

So we come to unfold rule. So unfold rule is very important for what? It's for to come up with the material length before you go to bend it, before you go to manufacture it.

So Inventor provided three ways to calculate the-- develop the length for the whole material. So the most basic one is KFactor. So KFactor is the factor to define the position for the neutral surface.

So from this image, we can find that KFactor equals gamma divide mu. So KFactor is a value from 0 to 1. It means the material from softer to hard. And a typical range for the KFactor is from 0.3 to 0.5. So really, you decide it based on your material.

So with this equation, the Inventor calculated that material length before you go to building. So from this, LPA is the burn allowance. It's the length for the neutral surface arc length. So with L equals LM plus LB and then plus LY, it should [? final ?] develop the length for the whole material.

So from this simple example, we can check that it's exactly the same value after flight pattern. And the value is equal with the formula. So just trust Inventor.

And bend table-- bend table you really got from your supplier or your manufacturer that you can import existing TXT file that in this bend table. And it defines a length correction value that is specified a bend radius and a bend angle. So internal, we are using this equation to calculate the total length. So it's much like the equation for deduction.

Later, we'll see this. So this is the last [INAUDIBLE] one in Inventor [? that ?] to calculate the length by the equation. So we have four types of equation-- bend allowance, compensation, deduction, and the variable KFactors. So you can confirm the image that the annotation for L1 and L2 is a little different. But it doesn't matter since you decide which equation to use is based on your manufacturing condition, your material.

So in each equation that you have the opportunity to define the formula is specified [? a ?] bend angle that you can edit each of [? mine ?] or add your own ones in the specific range. OK, so after you set up all rules, all styles, and then you want to share with the team. So you can save this file as a template. And then finally, you can use this template directly and everyone can use it.

OK, so here is a quick demo, Inventor. So this file, that is I created before the class. And now this is another access point to access the style and standard library.

So here, I created many Sheet Metal rules. But you have your own file name definition rule. So my rule is much simple with material and with thickness. But your rule must be much more professional.

So most important thing is that we need to define the thickness, define the material, and unfold rule is embedded in the Sheet Metal rule definition. So once you-- you can edit the unfold rule

by click this button. And it will go to the unfold rule for this Sheet Metal rule. So also you can change back and change another unfold rule.

So after you finish that, just save as a template. And in the template, file location. So I don't need to do that since I created one called [? AO2016. ?]

So next time [? I knew ?] a file that it's Sheet Metal, the new template will be shown here. So come back to the slides. Now we will go to the Sheet Metal features. So it's a [? list ?] for the Sheet Metal system.

So what we will learn? That in this election, we will go through some of the base features that how to create the Sheet Metal body initially and then the secondary features to continue the Sheet Metal design based on our base features. And also, another popular workflow is to convert Sheet Metal from a standard part to exact and extract a Sheet Metal plate from existing solids.

So with all these feature tools that you get the chance to make a multiple in Inventor Sheet Metal. So flat pattern is another important area we need to mention in this section. So there are four main tools in Inventor to create a Sheet Metal solid.

So I mention this because from 2016 that we provide multiple body support. And for these four features, there are options to say I want a new body or I want to join with existing bodies. So its a face, contour of flange, contour of roll, and a lofted flange.

So I add [? extrude ?] here. It doesn't mean it's not important since its a general modeling tool and you can use [? extrude ?] anytime you want to use. But just [? extrude ?] the required thickness. So with these [? basic ?] features that you have the opportunity to create the solids.

So in this example that there are nine solids in this sampler and each of the solid is studied with a base feature. So take this as a simple example that you can click the first profile to create a new solid. And for the second profile, then you can choose that I want to join with an existing solid.

And for the third profile, you say, I want a new solid. So after doing this, there will be two solids existing in this Sheet Metal part. So with the contour flange, so its like you must have a sketch profile. And you use this sketch profile to slip it through edges of [INAUDIBLE] with a distance to be normal to this sketch profile.

The most important thing is that during the creation, if you have multiple edges still acted, that auto [? mattering ?] will be created automatically. It will save many steps for you if you just use general motoring tools to do this work. OK, here it comes, the first [? tip. ?]

So for lofted flange, you can loft [INAUDIBLE] close the profile and it can be created successfully, but if we only we close the profile that you cannot do flat pattern. So before you do flat pattern, you need to create a rip to make it happen. So for contour rule, the tape is that you must create an open profile for this command to make it work.

So here is a very simple demo to connect all these base features. So I prepared some sketches in this model to save some time. So firstly, I created a sketch for [? fast. ?] So [? fast ?] is already created that with our new solid [? stalactite. ?]

For the second one, want to use the contour flange. So this profile is selected automatically. So now I have the chance to say I want to create the contour flange along these edges.

So I can pick these two edges or I can just pick a loop. Then I can creative contour flange in the center. And here is the option to say, do you want to apply [? automatic? ?] That by default, it's checked on. So if we check it off, it's not a requirement.

So we can create this one. And here is a chance to say, I want to create with my new solid or by join the existing solid. So let's pick again.

And now we go to say I want to create a lofted flange. And I loft from this sketch to this line. And here is some options to say that the four main methods for the lofted flange.

Press [? brick ?] is checked on by default. Also, there's another [? die formed. ?] So you can choose based on your own manufacturing needs.

So finally, we got a chance to create a contour rule. So for this contour rule in this sketch preparation that it's [? attendant. ?] So you can connect these two features together.

OK, so this is a quick [? rule ?] for these four basic features. But I want to mention is [INAUDIBLE] control flange that actually, they are very powerful always extends, [? too, ?] that you can use edges. Also, you can use the [? west ?] values to control the extent for this flange.

So we will mention it later, since this functionality is shared by many commands like flange, like [? harm. ?] OK, so come back to the slides. So for secondary features, that there are a lot of

secondary features.

But we will not go through everyone since the time limited. So we'll just provide some tips for many of them. So flange is the most often used secondary features to continue the Sheet Metal design.

So from this simple case, you will find it's two solids. So when you pick edges in flange, then it's supported to pick edges across the solids. So you can pick it is from different solids.

So [? automatically ?] will be created automatically. So that's powerful. And also besides flange, for corner seem, for cut, and bend, and it has the same functionality that you're all [? supposed ?] to pick edges to cross solids. But the request is these solids must have the same Sheet Metal rule.

So here is a tip for flange that sometimes that you want to create a flange to be parallel with a sad face. But the angle is not easy to find if you flange from the top edge. So it's easy for you to say I want-- you need to create the angle as a parameter.

And then you link this parameter in the flange angle. So any time you change this angle that the flange will be updated associatively. So I can show this model.

So for this model, it's easier to find this angle since this angle is created from the cut feature, cut sketch. So this angle is created. So in flange, it just links this to D16. So sometimes we use these methods.

If you want to link parameters from the features, you just click through the [INAUDIBLE]. And so for example, I want to link some parameters from this feature. I just click this feature.

And then I can pick dimension from the screen directly. So when we change this parameter to 60-- so this flange will be updated associatively. So for other cases that-- if there is not an existing angle to make use of, sometimes we will create a work plane to be a parallel with this face and then create a profile on this work plane and use face command under the flange to create this feature.

So next-- so next is cut. So it happens often that the cut is not normal to the target faces. So once you meet this situation that you are required to check on the cut normal. So when it's not normal, you check on Cut Normal. So it's very easy to remember.

AUDIENCE: Is there any way to set that as a default option?

NINA SHAO: Yes, it's default.

AUDIENCE: In [? node-- ?] so I think the default option is the Cut, like the [INAUDIBLE] Cut. So--

AUDIENCE: I know there's [INAUDIBLE] preferences [INAUDIBLE]

NINA SHAO: Oh

AUDIENCE: [INAUDIBLE]

NINA SHAO: I'm sorry about that.

AUDIENCE: [INAUDIBLE]

NINA SHAO: OK.

AUDIENCE: [INAUDIBLE]

NINA SHAO: I think we can recall this problem and you can find this in answer [INAUDIBLE] when you time. And we can practice it. OK, thank you.

AUDIENCE: So from my perspective, I think that this is not a general problem. So maybe we can submit the [? CR report to us ?] and provide the [INAUDIBLE] so we can have the future investigation and provide feedback to you. So if you don't have any question or problem, you can [INAUDIBLE] communication. Thank you.

NINA SHAO: OK, OK.

AUDIENCE: Isn't that setting persistent in the document settings [INAUDIBLE]

NINA SHAO: So [INAUDIBLE] ask if the last settings is saved. I guess it is.

AUDIENCE: I think it is. [INAUDIBLE] should be able to select [INAUDIBLE]

NINA SHAO: Yeah.

AUDIENCE: [INAUDIBLE]

NINA SHAO: Oh, got it.

AUDIENCE: [INAUDIBLE]

NINA SHAO: Yeah, yeah. I think it's a good idea, yeah. Yeah, It's a workaround.

But we will say if it's impossible to add options somewhere, since we have a bunch of options already. So people hate options. So we need to consider that, whether or not we need it.

Or just [INAUDIBLE] in this [? command. ?] So this-- another team's that sometimes, you need a cut on the road surface. So if you're just a cut, in this situation that you always will be filled [INAUDIBLE] fulfilled for the manufacturer.

So the recommended workflow is that if you already get this step and you need unfold, unfolded to be our planner. And then you cut-- you create a cart and then refold it back. And now you will get a beautiful result for the cut in the flat pattern. So for punch, you [INAUDIBLE] that you need to prepare two [? diskettes ?] firstly with a center mark, with center points, prepared.

And then go to click punch tool. And you will be navigated to the punch library. We provided some existing simple punch tools there.

And it's qualified data and you can [INAUDIBLE] it and then you can just put it on your sketch point like this, the dinner plate. And it's easily to create it. But if you don't have a punch-- you don't have the punch in the library that you want to need, then how to create it [INAUDIBLE] it's not [? usually. ?]

It's required that you create with a feature that we have five features supported at feature creation. So it's cut, extrude, revolve, sweep, and fillet. So you cannot use other features, or else that they will not be recognized by feature creation, by a feature publisher. So for this example, we create this punch feature and with many of revolve fillets.

And then we go to extract our feature. It's a publish. You can publish this punch feature to be in the punch library. And then you can locate in this plate [? easily. ?] So let's have a quick demo.

OK, so this is a sample case that we create. Firstly, there is a faceplate, and then we create a sketch for this feature. And the first delivery [INAUDIBLE] the first step. And then we get the second step.

So we get this window state in two steps. And now I go to the manage tab and find the extract feature. So for punch tools, [? then ?] you need to check on this Sheet Metal punch feature. Then you can choose these two steps and say this feature only includes these two steps.

And now you can specify the sketch. And now you can save it to the punch tool and you can see it in the punch library. This is already created.

[? Container ?] one. So to make it simple, just click center point. But actually, you need to make it seriously. So click punch tool, and I can get a [INAUDIBLE] size.

So I can pick this feature. Just say I want to put it here. So when you see the color becomes this one, it means that this ship cannot work for this plate.

It often happens that when the thickness [INAUDIBLE] the original one. So you need to change the size to be bigger. And after you refresh and click it-- and you can put this punch tool in this plate successfully.

AUDIENCE: So we're talking about [INAUDIBLE]

NINA SHAO: OK.

AUDIENCE: Is there a way to include parameters in punch ID so that when you go to detail on a drawing, for a slot, you can actually have a [INAUDIBLE] and width of that slot.

NINA SHAO: OK, So--

RIVER CAI: So [INAUDIBLE] so [INAUDIBLE] we use the punch notes [INAUDIBLE] the punch notes. So I think you have used the punch notes in drawing so you can edit the punch [? ID. ?]

AUDIENCE: Right, but if we have a slot that can change length and width depending on what you entered in for your options there, when you bring that information into that punch ID and display that [? drawing. ?]

NINA SHAO: So you want to display the punch parameter together with the punch ID.

AUDIENCE: [INAUDIBLE]

NINA SHAO: OK, we'll take notes. Thank you. So then it's a flat pattern.

So Inventor from the [? file ?] structure will have the two folders and [INAUDIBLE] them to

[INAUDIBLE] for Sheet Metal. So one is the unfolded model, and the other one is the flat pattern. So usually, we use flat pattern for two purposes.

The first one is to validate, to prove it's a validated design. So if the Sheet Metal can be flat patterned, [? then ?] it means that the Sheet Metal design is good. But if it cannot, then you need to recheck your design to make it work.

So only one exception is the [INAUDIBLE] feature is just a punch that-- for a feature that [? accept ?] [? cut through ?] features that for other punch features that they cannot be flat patterned. But it doesn't matter. It just has different presentation options for you to choose that you can only display the sketch point, center point.

Or you can display the sketch or just this [? unflat ?] patterned form in the flat patterned status. And the other property is [? that they're ?] for drawing and manufacturing. So as you mentioned that you want to create the punch table, punch notes with ID always parameters.

That's one request. And most of the [INAUDIBLE] [? open ?] table and the bend notes as listed in this image. So here is the [? tips ?] for flat pattern.

So normally when we do modeling in folded model, the flat pattern will be updated at the same time. But when your model becomes complex, you have maybe hundreds of features or you use [? a ?] logic to [? drive this ?] complex model. And all you have hundreds of patterns, pattern instance, that then it will make the updates to be slow, to make the flat pattern updates to be slow.

So there is an option called differ update. And you can check it on. And when you check it on, the status, the icon sign, will tell you that it needs to be updated but it's [? held on. ?] So you can do it later.

RIVER CAI: Just one comment. So if you open the [INAUDIBLE] you can select this option from [INAUDIBLE] So we have a complex shipment of parts, these flat patterns. So if [INAUDIBLE] we can take this option [INAUDIBLE]. So maybe the performance should be [INAUDIBLE]

NINA SHAO: OK, OK. Good comments. So it's easy if you just open a large Sheet Metal design. So you can make it quicker from the very beginning.

So this is a popular workflow that-- but this image showed a workflow that you get some Sheet Metal metadata from a non-uniform solid. So this is a solid, and I want to get a Sheet Metal

design based on this solid. So [? really, ?] there are two general tools to use to extract the sheet plate from a solid.

One is shell and the other one is a thicken/offset. So it's popular used. So when you click the convert to Sheet Metal from standard part-- so here is a tip that many people may ignore this [? true ?] tip that says [? inside, ?] select base face.

But since the information is not complete, so it's easy to be ignored. But it means that you select best fit and the Inventor will get the thickness value directly. So you pick a face and we get the thickness.

And we'll fill in the Sheet Metal default thickness override value directly. And then for the all. Downstream features, we'll follow this thickness definition. So we go to our quick demo, how to make it.

So for this workflow, it happens that I extract the sheet information before go to [? conversion ?] Sheet Metal. So in this demo, I will show that a quick one. It's like how to use a shell to do it. So I use 0.1 inch for this shell creation. And then when I go to convert to Sheet Metal, here is [INAUDIBLE].

You will see select a base face. So I can pick this face. I think I get a defect. Sorry for that.

AUDIENCE: [INAUDIBLE]

RIVER CAI: So because of-- if you select-- maybe sometimes, it's better for us to choose the select. Most biggest [INAUDIBLE] to detect the thickness. So you can see that [INAUDIBLE] select the position close to the corner. So maybe it's something wrong here.

NINA SHAO: Yeah, I did a good demo for a bad case. So I do it again. So convert to Sheet Metal and pick this big face. Oh, sorry. I did it.

RIVER CAI: [INAUDIBLE]

NINA SHAO: OK. Whoa, get [INAUDIBLE] thank you.

AUDIENCE: [INAUDIBLE] the end. So what did you have the Sheet Metal [INAUDIBLE]

NINA SHAO: Yeah.

AUDIENCE: [INAUDIBLE] in my mind, you wouldn't want to change the [INAUDIBLE] these are the styles

you have in your library that match it--

NINA SHAO: Sure, sure.

AUDIENCE: Because you want to use the correct styles [INAUDIBLE] correct [INAUDIBLE] format [INAUDIBLE]

NINA SHAO: Yeah, so River, answer this question.

RIVER CAI: So I think i got your question. Actually, you have [INAUDIBLE] the [? template ?] [? instruct. ?] So you share the [INAUDIBLE] match the [INAUDIBLE], you defend, then convert part to [INAUDIBLE].

OK, so if you have defended the [INAUDIBLE] rule or [INAUDIBLE] in the template, [? then ?] you share the body, share the part. You can use the thickness from your defended [INAUDIBLE] at the [INAUDIBLE].

NINA SHAO: Yeah, so--

RIVER CAI: Because this information is solid in the [INAUDIBLE]. So you can use the thickness here at the distance.

NINA SHAO: Yeah, so like, remember [INAUDIBLE] that one you convert to Sheet Metal and you can use anything from the Sheet Metal rule. So I just tap the thickness in this dialogue and it will use the thickness from the Sheet Metal rule directly. Yeah.

AUDIENCE: [INAUDIBLE]

NINA SHAO: Sure.

AUDIENCE: [INAUDIBLE] So let's say you [INAUDIBLE]

NINA SHAO: Yeah, thank you. So when you are talking about this, I'm just kind of [INAUDIBLE] to share something about the new features. But it's something related.

So if you are really interested in this thickness definition for multi-body, [INAUDIBLE] you are encouraged to involve us in the beta [INAUDIBLE] [? room ?] to test with a new tool. So next, to make the demo simple, so just to say I want to create some opening. Create a rip for these edges.

And then go to [? bend. ?] Say, to make it to be a Sheet Metal part. It's Sheet Metal now. And to validate if it's qualified, we could create a flat pattern.

And it's successfully. So this one is good demo. OK.

OK, so this is the main purpose for this class is to introduce top down design. So we want to start with the Sheet Metal rules and the Sheet Metal features. And we know the capability, the basic capability, for Inventor in feature level-- that for features, that you can create a [INAUDIBLE] some of these just the functionalities. But how to make the design is a little different. So it needs the method and a methodology and to connect these tools together and to make your work easier.

So firstly, we want to summarize the key characteristics of top down design. So it's a method and it has some key factors like it's in contact [? to ?] design. So its design is [INAUDIBLE] in the same context.

So the second one is in support for collaboration since everyone shares the same picture. So it works for a team and to share the work. And the third one is it's high efficiency on design changes.

So since [? you're ?] [? really ?] top down design have a skeleton model as a basic, so any changes happened around the key, around the skeleton model, that it will be updated, go to the downstream directly. And the last one is a quick response for cost control since the design change is easy. So it's much easier to make quick changes based on the cost.

So the image side [INAUDIBLE] architecture world. For the architecture world, it's usually top down design, have a whole picture and a split, and go to a different department and then combine together. But usually for bottom up, then we have a typical case is like children play a Lego game. And it plays one piece by piece. And it's bottom up.

So here is a model created. I hope it makes sense to you. So for top down design, it usually comes from the [INAUDIBLE] after design or bone structure design. So you have a clear picture on what the overall design will be.

And so the layout is in the center. And based on the layout design, you can extract some skeleton model. And then for each component, you need to define the reference geometry and the metadata for each component to use, to link. So for component 1, 2, 3, 4, any component, they are all associated with the layout.

So for all these piece component, they just come up with a whole picture for the product. So that's the method for top down design. So currently, we have three that really use the top down design approaches. Its approaches to achieve top down design [? with. ?]

So the first one is to new [? parts ?] within assembly. So this is the most basic one. It's just a new assembly, create-- import the skeleton model. And then you add the new parts based on the reference on the skeleton models.

And the second one is use derive, that you create a new part and then you derive a skeleton part model in your part. And the third one is our recommendation, is the Sheet Metal multi-body. You use the Sheet Metal multi-body to create the contacts in one part and then make components to make assembly.

So go to details. So from the file structure that you're ready for the first one. It's-- I only gave examples for simple case. So you can imagine that how to use it in a big assembly.

So assembly 1-- and always a skeleton part in. And you create a part 1, 2, 3 and by geometry reference on the skeleton part. And this one is assembly 2 and you have different part.

But each part is derived [INAUDIBLE] a skeleton part inside the part. And this is a multi-body work flow. And from the flow, it looks compact [INAUDIBLE].

But you can see this is the result-- that by comparing the file management, it has one file less than other method. So it goes through them one by one. The first one is the parts with assembly.

So for this part, this blue one is a skeleton model. And I create assembly, place it in, and create a new part which is associated with this skeleton model. So we can make a quick demo for this one.

So you can hide the existing one. So I create assembly and place this skeleton part in. So then I create a new part, maybe part two with this new template.

So here is a [INAUDIBLE]. I guess maybe it's not a new tool [INAUDIBLE] that when you create a new part in assembly, it's recommended that you click the center [? print ?] in assembly to create this part. Then it will make each part the coordinate system will be sending its part with assembly.

So now I create a sketch to project the top face for this model. And then I use face to create the first shipment feature. And now we will try to say I want to create these [? phalanges ?] and locate that how to make it associative.

So I want to make this associative with the skeleton model and I need to choose two geometry. But there's nothing to pick, right? So before that, you need to create a work point. Let's [INAUDIBLE] the [? border. ?]

So a work point here is created. It's adaptive. So it's associative with the skeleton part.

And now when I got back to the flange, I can choose this work point. So apply. And I can continue to create flange. And this time, it's just a distance. And with flange, it's very easy to change the angle.

And next we will say why we say it's a mandated to use the center point to be-- the center point of the part. And now I want to mirror these features in this side and in this side. For this model, that-- whoops.

I can choose these three flanges and to [? be mirrored ?] with this. I can create it directly. And then again, I can pick this features.

[? See, ?] I can along this work plane and then create another one. So it's easy to reuse your design in one part quickly. So I will [INAUDIBLE] more about the flange that-- about the [? vise ?] [? extend ?] functionality. so if I want to extend this flange but with wider width-- so how to make it.

So by default, it's with width extend methods, it's based on edge. So how wide are the edges that how wide the flange will be. So we can choose width from this dialog and see if it's auto-centered.

So I can just say-- whoops-- I can measure this length [INAUDIBLE] too small. I'm [? picking ?] the larger. OK, and this one will be bigger. Also, there are other options like offset.

I can offset from this length as a five millimeter and another as, what? As five millimeter. So we can chance to see the difference.

So it's very flexible then for you to define the width for the flange. Also, this option is used in control of flange and the [? hem. ?] So it's the same experience.

And now we just do these steps. It's enough. So we go to the skeleton part and show this [INAUDIBLE] and change the width to be five inch.

Do update. OK, I need to go back to this. So you see this part has been updated associatively with this skeleton model, right? OK.

AUDIENCE: Good job.

NINA SHAO: Thank you. [? Remember. ?] So let's talk about the advantage or disadvantage for this method. So we say that everything happens in one context.

So I can refer the geometry info from the skeleton model by creating some mediums-- so by creating some adaptive work plan, adaptive sketch, adaptive work points. But the disadvantage is that these adaptive stuff, maybe it's not so reliable that [INAUDIBLE] its [? break ?] that you need to take some efforts to save the relationship back. And another bad thing is that when the assembly becomes complex, since this method requires you to always work in context with in place editing status, you cannot just open a part to do the job.

So this will bring some problems if the assembly becomes larger. So let's go to another one. This one would be better.

[? But ?] this one is use derive, [INAUDIBLE] derive the skeleton model inside of this part. So the [INAUDIBLE] derive that derive has many different [INAUDIBLE] derive with surface or derive with solid. So what's the difference?

So a derive with surface-- that it's [INAUDIBLE] surface body in this Sheet Metal part. And usually, that you can use [? thicken ?] to make the Sheet Metal features quickly since I can just directly thicken this surface to be a new one. And another thing is that you can just do flat pattern in this part.

So with currently, the existing multi-body supported in Inventor that if they are multi-body in one part, that it's required to make parts for each solid and then to do flat pattern. So for surface derived, that it don't need extra steps to do that. But the problems is that when you want to refer the geometry information, that from the surface body, it's also required that you create some mediums [? there ?] to connect them.

So I will do a demo later after talking about the solids. But for solids, that you can derive this skeleton model as solid in this part. The benefit is that for solid-- and you can take the [?

vertex ?] directly for the flange equation to connect these two geometry information together.

But for this workflow, you need this to support. It's like you need to make part to make this solid to be a single part and then to do flat pattern. Then this part will be associative with this part.

So any changes from the skeleton model will go to this part and then go to this part. So it's quite straightforward. OK, so let's see how it works.

Need to run faster since the time is-- so this one is already created. So we can check the feature. It's designing [INAUDIBLE] side is just the same skeleton model.

So when the skeleton model changes, I get a notification just to click the button to update the model. So for this site, [INAUDIBLE] OK. I derive this model as a [? surface. ?] And the first step, I just thicken this surface up. And for this, also I use thicken.

And I create a [? bend ?] to connect these two bodies together. And continue the flange. And others is the same. We can just use a mirror, same as the first model to do it.

So for surface derive, [INAUDIBLE] most often used tool is the thicken. And then you use a blend to connect these two bodies together. And how about the solid one?

[INAUDIBLE] get a notification, [INAUDIBLE]. So for solid, it could be very easy, since I can just thicken the first one as a new solid. And then when I do the flange, I can choose the vertex directly as the height extends definition.

So I just do this. I don't need to define extra geometries, extra work geometries, to connect the relationship. OK, so now that I needed to do one step, make part. I make a solid too to be a part.

And I can say I want to place it in an assembly or not. This time, I choose not. Then I create this part. And now I can do create flat pattern and to check the design. So that's the work flow from the multi-body to make part.

AUDIENCE: [INAUDIBLE]

NINA SHAO: Yes, the Sheet Metal rule is linked. So I will show you here.

RIVER CAI: So I think I [? got ?] your question. So you mean the derived shipment part-- shipment rule is

linked to the [? source ?] part, right? So--

AUDIENCE: [INAUDIBLE]

RIVER CAI: Yes, yes, yes, 2016. So if you change the Sheet Metal [? stages ?] in Sheet Metal rule, these changes can be synced to the derived shipment [? of parts, ?] yeah.

NINA SHAO: Yes, so look at this option here, that there's linked Sheet Metal [? stale ?] option. That means that the [? default shipment ?] rule will be transferred to the downstream, yeah. OK, come back.

AUDIENCE: [INAUDIBLE]

NINA SHAO: Oh, sorry.

AUDIENCE: [INAUDIBLE]

NINA SHAO: OK, it's a good question. Then--

AUDIENCE: [INAUDIBLE]

NINA SHAO: Yeah, it's something we are working in progress. And so we [INAUDIBLE] to do the validation for the new functionality. OK, so here comes a typical multi-body design.

And for this design, it's from scratch. So from scratch means I don't need to prepare a separate skeleton model. This is just the skeleton model.

Its a skeleton model for everyone. The [INAUDIBLE] reference each other. So its created from the scratch mostly for simple design.

So for this [INAUDIBLE] glasses and this cover is the first created. And for this two part, it's created based on the first body. So there are three bodies in one part.

So with this design [INAUDIBLE] similar workflow that you [INAUDIBLE] multi-body in one part. And then you use make part. You can use make part to make each body to be a single part.

Also, you can use make component to make select your solid to be parts in your final assembly file. So that's the top down design from scratch. OK, let's see it.

OK, for this part from the file structure, we say there's a cover, there's a holder, and there's a support plate. So [? every ?] solid is created in the same context. So [? they ?] self reference

[? that-- ?] the geometry information.

So firstly, I create this cover, quickly go to this one. First, I [? create ?] this body and with geometry information. Then we go to create this face.

Since there is a fixed length, fixed distance required from the end face of this [? cover ?] to this face, right? So we just reference this face to create our work plane to place the second solid. And the same situation for the third solid, and it starts from a face here.

And then you can do flanges and finally to make it. So with these three bodies solid in one part that you really will be using the components. And we can choose also three solids in this design.

Also, it depends on your needs. And you can just pick some of them. And go to Next.

And you can see here it also has a link Sheet Metal still option. I didn't check on color override. So it [? removed ?] colors.

But we can [? see ?] the color back by check on the use color override from the source assembly and from the source part-- sorry. So when we go to each part, that we can do flat pattern for each one. So during our design, you can make it at the very beginning that when you get a basic shape for this solid and you can create-- make components or make part to validate your flat pattern anytime you want to.

So I need quickly-- so this one is the multi-body design from scratch and the final one. This one is Sheet Metal multi-body from skeleton model. So this is a method from the packing industry that really you [? got ?] a [? center ?] material and [? a ?] [INAUDIBLE] is created based on the [? central ?] solid.

So you get the right size. When this central solid changes, all the [? colors ?] in this part will be changed together. So in this example, they are five solids in one multi-body and all created based on skeleton solids.

AUDIENCE: You we import solids from SolidWorks or other programs?

NINA SHAO: Yes, yes. So-- oh, I missed these tips. So when you want to impose maybe Sheet Metal design from SolidWorks or other competitors, that Inventor cannot recognize its Sheet Metal part directly. So there's one quick way to rebuild it quickly. It's like we can import it in and use

[INAUDIBLE] to quickly rebuild it and bend. I demoed a little bit on that.

AUDIENCE: So I [INAUDIBLE] those two [INAUDIBLE] multi-body or single body from third party software.

AUDIENCE: [INAUDIBLE]

AUDIENCE: [INAUDIBLE]

AUDIENCE: Bringing in full assemblies, and we get machines for our customer. And we have to [INAUDIBLE] do all the Sheet Metal [INAUDIBLE] we pull that in and we extract geometry [INAUDIBLE] in a lot of cases [INAUDIBLE] and we got to do [INAUDIBLE] bodies and do our [INAUDIBLE] parts.

AUDIENCE: OK. So actually, we can [INAUDIBLE] single body, multi-body [INAUDIBLE]. So you can import the third party [INAUDIBLE]

AUDIENCE: And then when [INAUDIBLE] design process [INAUDIBLE]

NINA SHAO: Well, yeah. Imagine a cool functionality Inventor. It's not in the area for this class, but it's the Inventor stuff-- [INAUDIBLE].

AUDIENCE: Yes.

NINA SHAO: Yeah, so you can do it and import third party model and keep all association with the source file.

AUDIENCE: Yeah.

NINA SHAO: Yeah, OK. OK, so this is the same workflow as the one from scratch. And when you make the multi-body design in part and then you use main components or make parts to make them to be single Sheet Metal part and do flat pattern drawings in this workflow.

So [INAUDIBLE] 15. So I think we don't need to demo this one either-- [? not ?] either. I don't need to demo this one since many approaches I just demoed. So I have a skeleton model provided.

So I think I can upload it in the [? AO ?] class and you can download it later to practice. OK, so we mentioned many times down the new improvements in 2018. So I think that it's a good opportunity to introduce a bit of [INAUDIBLE] to our [INAUDIBLE] and you can join us to test the [INAUDIBLE] earlier product in progress.

And then you can get a chance to provide feedback and join us with the design work. OK, that's the objectives quickly. And go to [INAUDIBLE].

Don't go. They're gifts. So I prepare some more questions.

So I give you some minutes to look at it. And then-- OK. So say it's a gifts time.

So ask your questions and [INAUDIBLE] I will stay [INAUDIBLE] go. And then you can raise your hand and answer question, OK? So the first one is can we make a contour rule with a closed the profile?

Go. Oh, it's so quick. Oh, no?

Oh yes. OK. Yeah, yeah, yeah. OK, so--

AUDIENCE: OK, so [INAUDIBLE]

NINA SHAO: Yeah, yeah. It's available since it's gift from AU channel. So you'll have double AU.

The second one is which option do you need to check on when the [? cart ?] is not normal to target [? faces. ?] Go. Oh, it's-- OK.

AUDIENCE: [INAUDIBLE]

NINA SHAO: Cool. OK, thank you. So next time, I say go and then you raise your hand.

OK, the last line is which features are supported to create a punch [INAUDIBLE] feature. OK, [INAUDIBLE]. You can just talk some of them.

AUDIENCE: [INAUDIBLE]

NINA SHAO: Cool, and the last one is cut.

AUDIENCE: [INAUDIBLE]

NINA SHAO: [INAUDIBLE]

AUDIENCE: [INAUDIBLE]

NINA SHAO: I think I need to spend more. Do you agree? OK, so I pick another thing.

Is [? there-- ?] OK. So what's the first step that after you click can work to Sheet Metal? OK, here is [INAUDIBLE].

AUDIENCE: Second [INAUDIBLE]

NINA SHAO: Nope. [INAUDIBLE]

AUDIENCE: Second [INAUDIBLE]

RIVER CAI: Yeah you're right.

NINA SHAO: OK, and pick a fifth. Pick a [INAUDIBLE] [? fifth ?] to get the thickness. Yeah.

OK. OK, so when the Sheet Metal flat pattern updates become slow, which option- I didn't say go! So which option you should check out? Go! Yeah.

AUDIENCE: [INAUDIBLE]

NINA SHAO: Yes, cool. OK, the last one very quick. OK, so for flange, that when you want to create a flange to be parallel with a [? sad ?] face, then what [? you ?] should do? Go. Back, yeah.

AUDIENCE: [INAUDIBLE]

NINA SHAO: Cool, OK. Thank you. OK, that's all tips.

And you can review the slides later. It's all online now. So I don't think we have more time for Q&A, but--

AUDIENCE: [INAUDIBLE]

NINA SHAO: You can give some-- one, 15 minutes or 10 minutes to-- 10 minutes and we have five minutes for Q&A maybe.

RIVER CAI: Any question?

NINA SHAO: Yeah, or comments. OK, then. Please.

AUDIENCE: [INAUDIBLE]

NINA SHAO: Oh, you mean--

AUDIENCE: [INAUDIBLE]

NINA SHAO: You send the file from 2016 into 2014 and you--

AUDIENCE: [INAUDIBLE] that can be [INAUDIBLE]

NINA SHAO: And get the Sheet Metal rules.

AUDIENCE: [INAUDIBLE]

NINA SHAO: OK.

AUDIENCE: [INAUDIBLE]

NINA SHAO: OK yeah I think it's--

AUDIENCE: [INAUDIBLE]

NINA SHAO: OK, people are [? hurried ?] for keynotes. OK, so for people who have questions, you can stay here. But for people who will, remember that you can find us in the [? answer ?] bar.

It's just besides the exhibition hall. So it's easier to find us, OK? Thank you