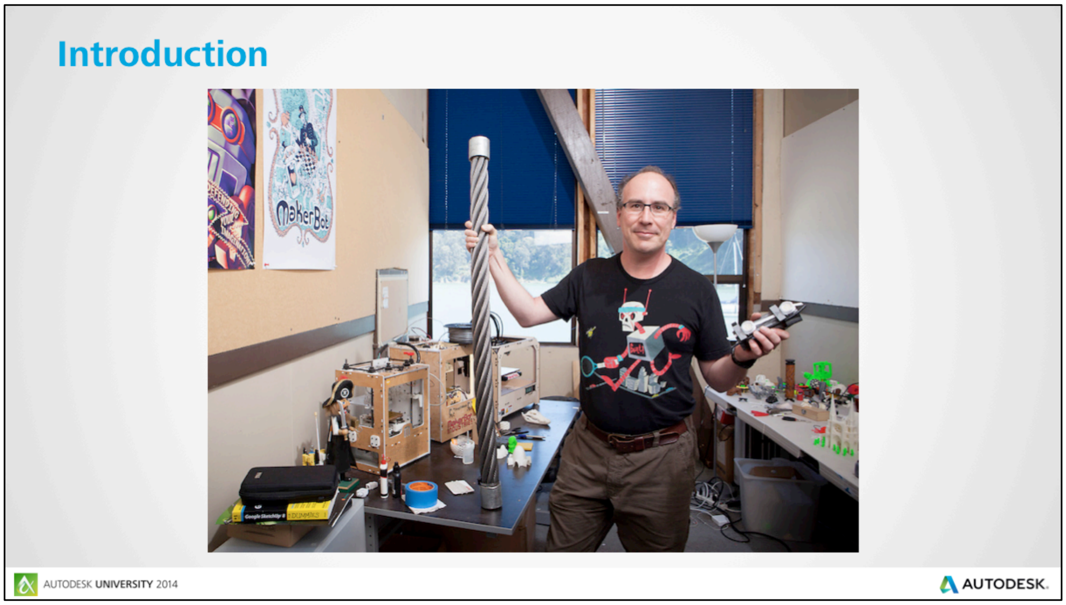
Design for Consumer Level 3D Printing

Gian Pablo Villamil
Product Line Manager, 123D
gian.villamil@autodesk.com

Agenda

- Introduction
- Goals
- How inexpensive 3D printers work
- Constraints and design tips
- Tools

AUTODESK UNIVERSITY 2014 AUTODESK.



Hi, I’m Gian Pablo Villamil

I’d like to start with a little about me, and what is relevant to the class.

I’ve been working with consumer/hobbyist 3D printers for 5 years now, mostly with Makerbot and similar machines. I have 3 printers of my own (3 generations of Makerbot), and experience with many others, including Type A, RepRaps and professional level Objet and Stratasys printers.

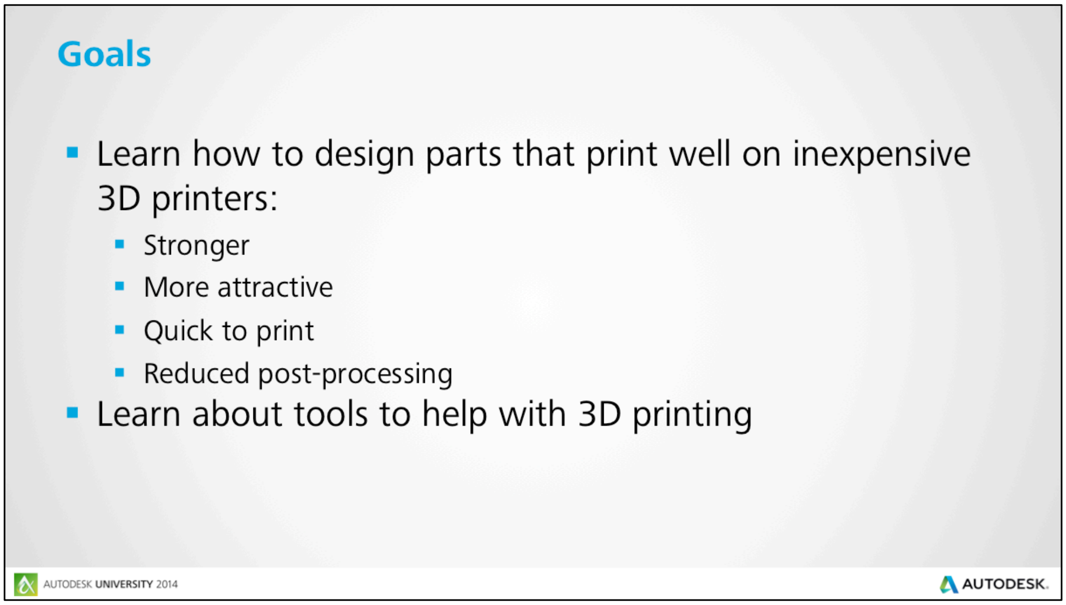
I’ve gone through 100s of lbs of ABS and PLA, and in the process seen these machines evolve from hobby projects to really practical tools.

User testing

AUTODESK UNIVERSITY 2014

AUTODESK.

My team has grown!

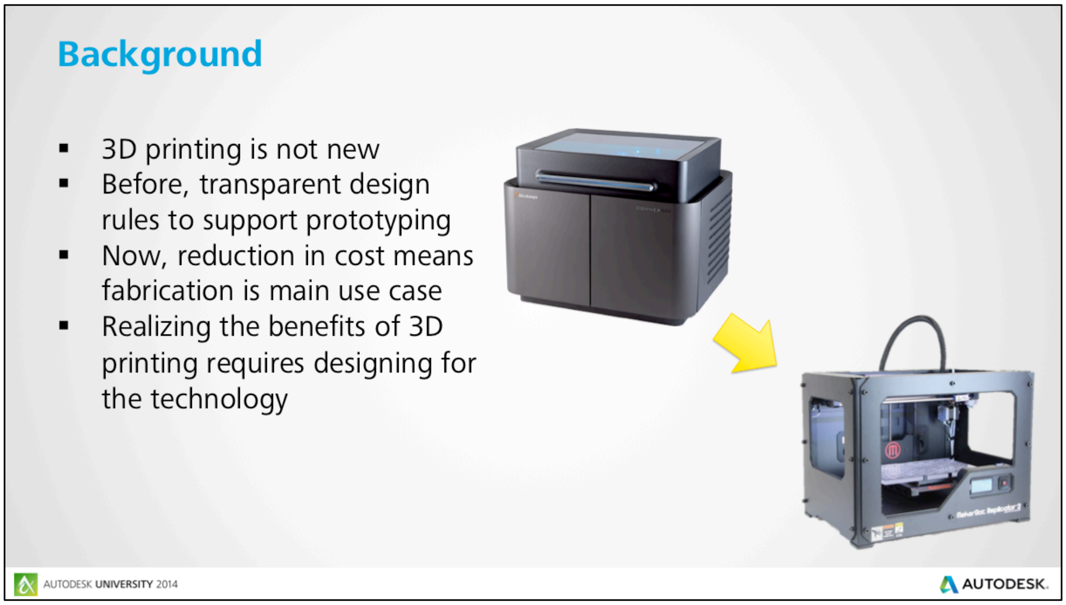


The goal of the class is to learn how to design parts that work well with inexpensive 3D printers – I'll talk about why this is interesting and important

We want parts that will be:

- Stronger – they can stand up to daily use
- More attractive – they look nice while in use
- Quick to print – print fast and use less material
- Reduce post-processing – can be used right out of the printer, or with minimal cleanup

- I'd also like to talk about some tools, from Autodesk and others, that are very helpful when working with inexpensive 3D printers



3D printing is not brand new – it's been around for quite a while, in various guises. Fused deposition modeling, the basis for most inexpensive printers, has existed since 1989.

So what's changed?

Early 3D printers were expensive, and focused on high value applications, such as prototyping. In order to be effective for prototyping, the design rules had to be as transparent as possible, so that a designer making a part for injection molding could ignore the constraints of the printer itself.

As 3D printing gets cheaper (both printers and materials) they are increasingly used for making parts for end use.

Since the parts are not intended for another manufacturing process, it makes sense to optimize their design around the constraints and design rules of the printers.

Parts for use: the emerging 3D printing use case

AUTODESK UNIVERSITY 2014

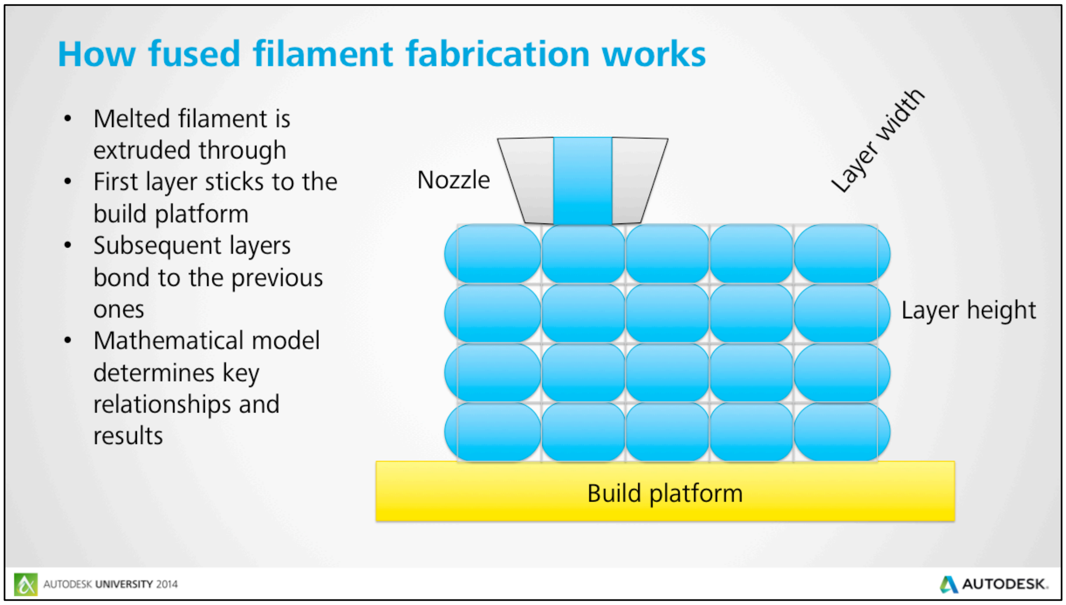
AUTODESK.

I'll talk about how these printers work – it's a necessary basis to understand where some of the constraints and design rules come from.

Be aware that you working with a combination of software and hardware, and many of the key parameters are set by the intersection of both working together.

I'll talk about the software, what it does, and what it defines, and then get into some detail about the physical process itself. I'll focus on Fused Filament Fabrication, since this is the technique used by most consumer level printers. (There are other emeging techniques, and I will address their constraints as we go if there is time)

Finally, I'll talk about how FFF leads to some constraints and challenges.



This is what is really happening:

The software calculates the volume of filament needed based on the layer width and height that you define. You also tell it the diameter of the filament going on, so it can make a calculation of how many mm of filament going in are needed to produce a deposited filament of given dimensions. (Incidentally, accurately measuring the diameter of your filament is key to obtaining good results – it tends to vary quite a lot, between 1.6mm and 1.8mm)

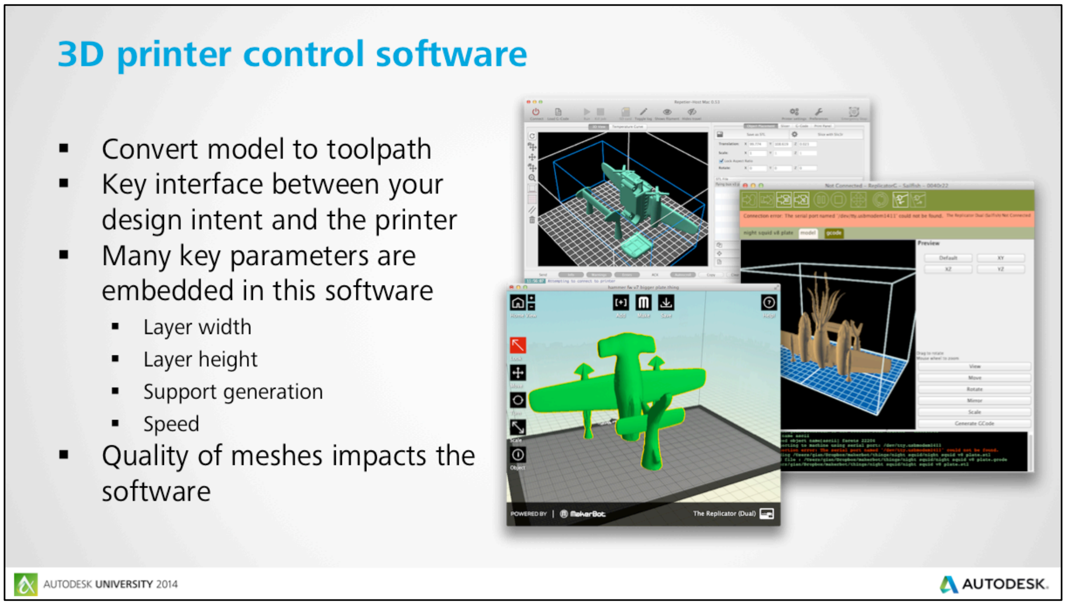
Now, the software model is not entirely accurate, and there are a couple of “fudge factors”. The grey grid represents the theoretical output, a square matrix. The blue represents the plastic that is really laid down. As you can see, the round extruded filament is “packed” in by layers below, to each side, and the flat sides of the nozzle. It doesn’t really fill in the matrix exactly, so most printer software has a parameter called “packing density” to adjust for it – lower the packing density to extrude more filament.

The other fudge factor is on the edges, where the extruded filament is not constrained on the sides – it bulges out of the grid. If you’re worked with professional printers and are coming to consumer printers, one of the things you’ve probably noticed is that the mechanical tolerances of the parts are a bit off. Increasingly, the printer control software is adjusting for it, but it is not quite there yet.

Note that for a strong print, you’ll want the filament to be as tightly packed as possible, but then you will have a greater error in the tolerances. Both of these parameters need to be adjusted in sync.

Finally, you want to be aware of the layer width – this is defined by the software, and is used in the volume calculation mentioned earlier. Ideally, you want it to be close the nozzle diameter, but it doesn’t have to be that way. The nozzle diameter simply defines a range of possible widths, if you go too wide or too narrow quality will suffer but the software will let you do it.

On Makerbots, the layer width is constrained to match the nozzle diameter. Other print software defines a ratio between layer height and width (usually 1.4)



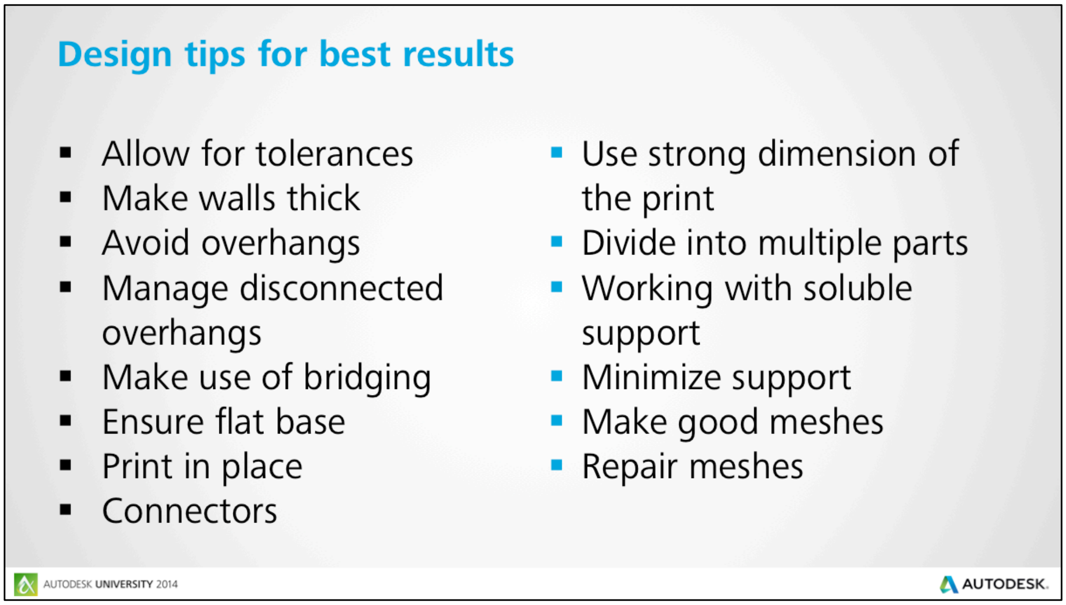
3D printers are a combination of hardware and software working together.

The printer control software has a key role in taking your 3D model and converting it into insructions that your printer can then follow, a process called slicing. So, how the printer software interprets your 3d model is a key factor in achieving good prints! It's the key point where what you mean to make gets translated into physical reality.

The printer control software is where several key parameters are defined, such as layer width and height, how support structures are generated, and how fast the printer moves.

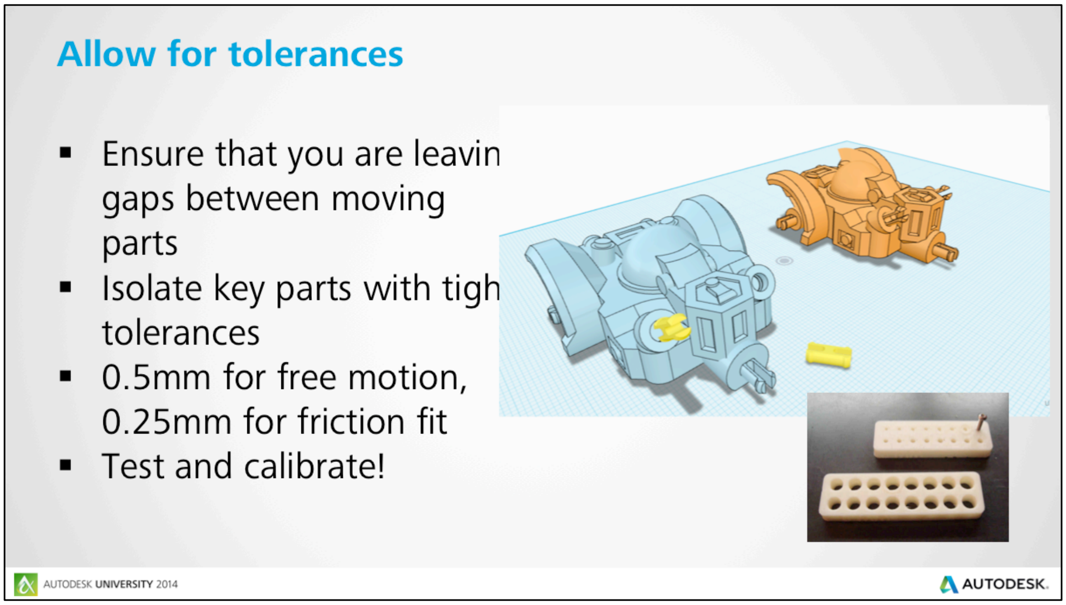
It's at this stage that the quality of the input mesh makes a difference.

Now it's time to talk about some design tips for getting optimal results



Now that we know the basics of how these printers work, let's figure out how to get the most from them.

I'll be combining a discussion of constraints with techniques for overcoming them.



If you are creating models with multiple mating parts, you need to allow a gap between them. (This addresses the inaccuracy we just discussed). Typically 0.5mm will give you a loose fit for rotating or sliding parts, while 0.25mm will give a friction fit. This will vary by printer, so you should print out a test piece like the one in the picture and figure out the tolerances for your own situation.

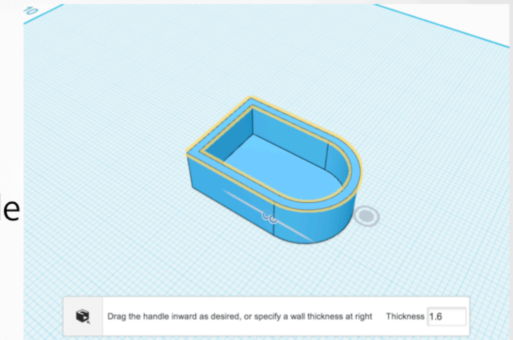
One key design principle is to try to isolate key parts with a tight tolerance. For example, the picture shows a snap together toy car. The original design (in orange) had the mating pin designed into the body of the car. The revised design (in blue) has a socket in the body. That way, if the tolerances are wrong, you only need to adjust and re-print the pin (shown in yellow). There is another benefit to isolating the pin that way, which we'll get to in a bit.

Allow for tolerances – pins & wheels

AUTODESK UNIVERSITY 2014 AUTODESK.

Make walls thick

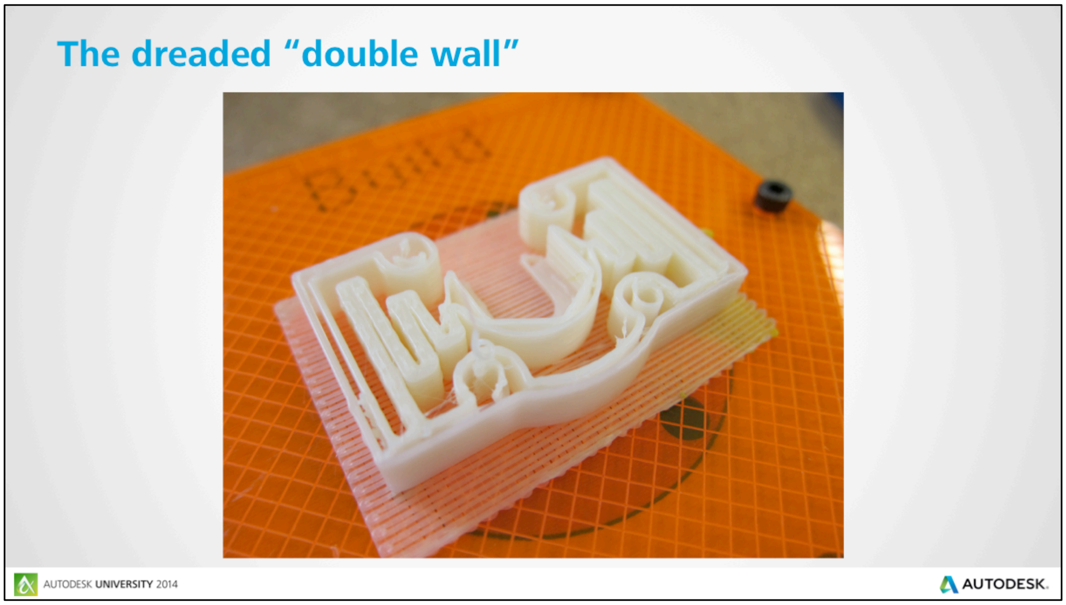
- Make narrow walls an exact even multiple of layer width, if this is less than # of shells
- Avoid the dreaded “double wall” problem
- Get the layer width from slicer parameters



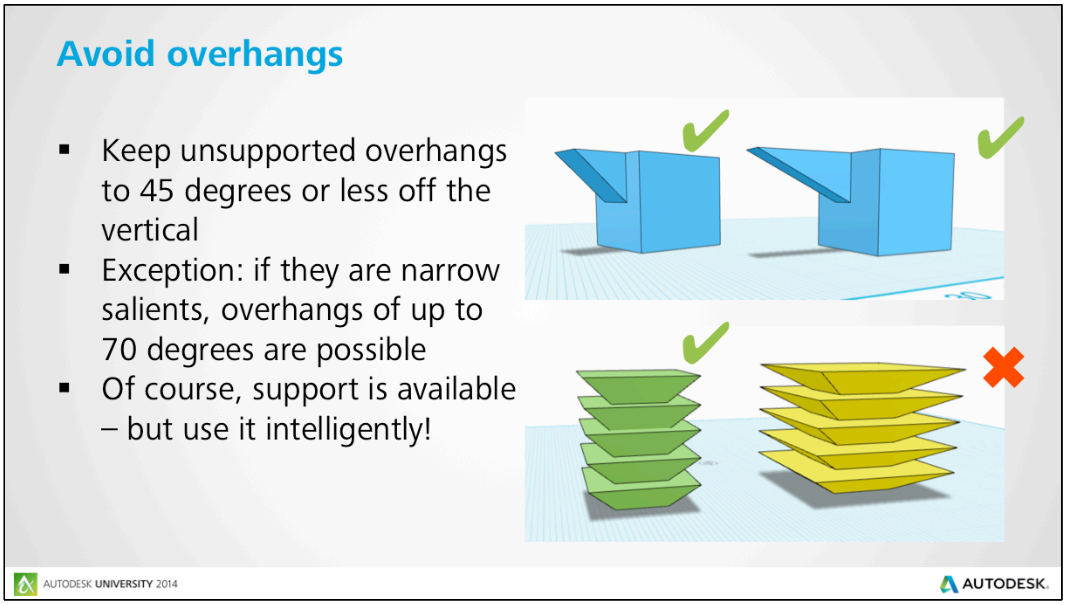
AUTODESK UNIVERSITY 2014 AUTODESK.

If you are making boxes or enclosures, you should make sure that the walls are an exact even multiple of the layer width. The reason for this is that the slicer has no way of accounting for very small gaps, and will simply not extrude anything.

So for example, if your layer width is .4 millimeter and you have a wall in your model that is 1.0 millimeter wide, the slicer will generate two walls with a .2mm gap in the middle. You would want your wall to be either .8mm or 1.6mm wide to avoid this. (Odd numbers don’t work either – though this is eventually fixable).



Double walls appear when the wall thickness is not an even multiple of the layer width. The slicer doesn't know what to do with the missing area.



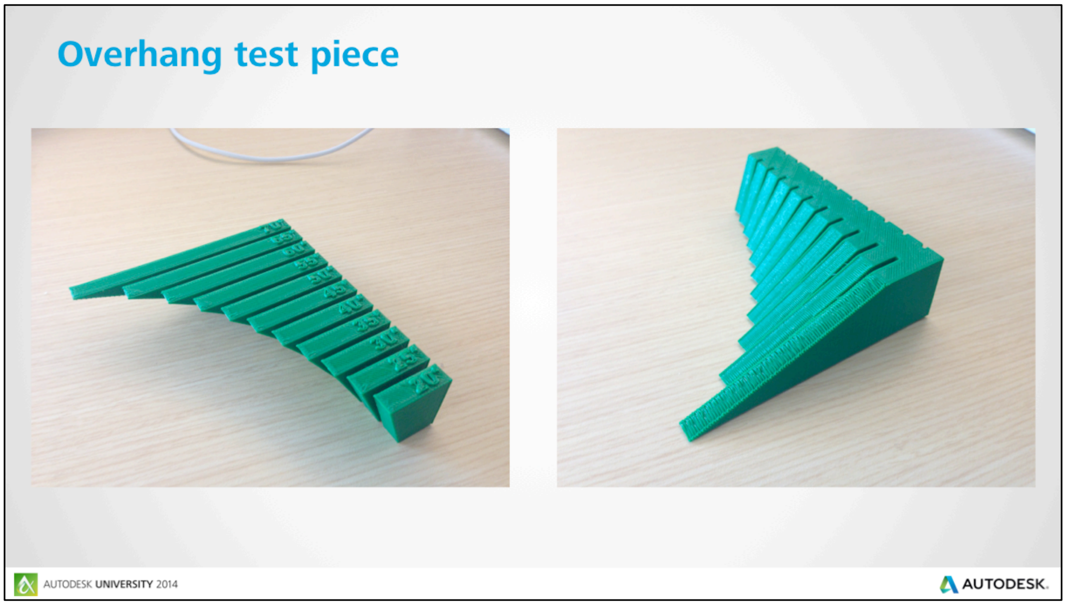
Overhangs are one of the best known constraints in FFF printers.

Since the layer width is greater than the layer height, the deposited filament has a rectangular section, and overhangs are possible. Generally the plastic is “sticky” enough to easily allow overhangs of up to 45 degrees off the vertical.

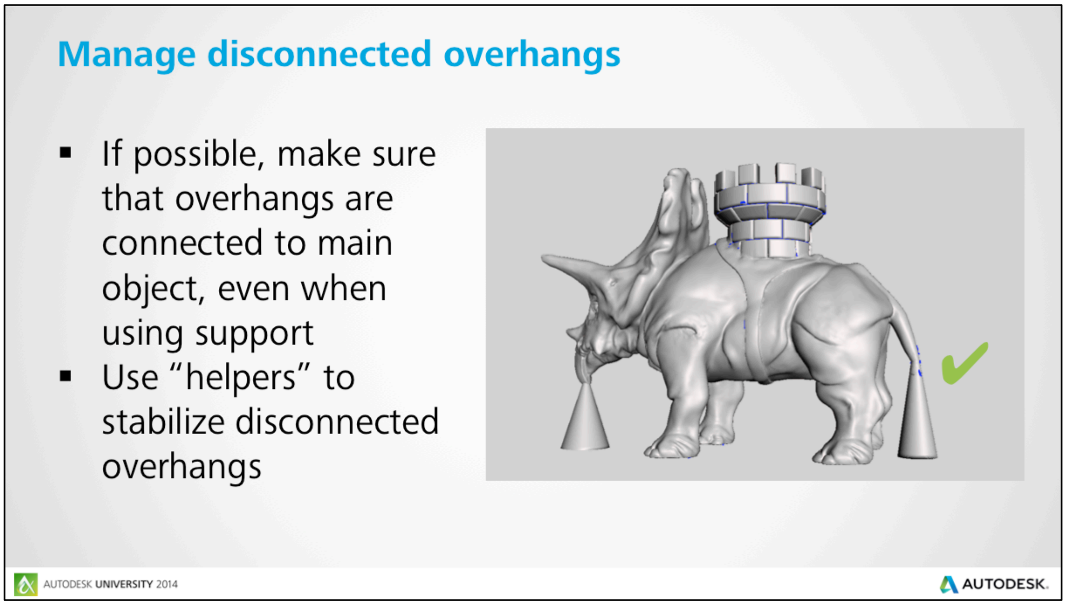
However, you can go even further, up to 70 degrees, if you have a narrow salient. The side walls are supported well, so you are in effect “bridging” across the bottom surface.

If this is NOT the case, the 45 degree rule applies. The green tower will print fine, but not the yellow one – the extruder would be printing into air.

Of course, support material is avallable and works well in this case – however, it will increase your print time and affect your surface finish.



Here’s an overhang test piece – you can see that it printed up to a 70 degree overhang, but when you look at the bottom, you can start to see irregularities

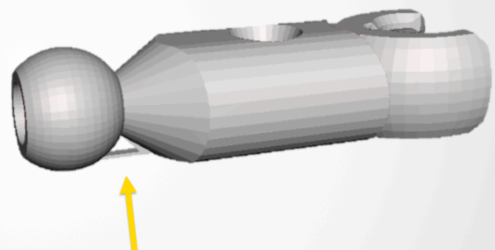


In some cases, you'll have overhangs that are disconnected from the main body of an object. In these cases, you want to design in support for them, even if you will later use the slicers own support generation tools.

The support generated by the slicer is intentionally "flimsy" and won't properly hold up structures like the nose and tail of the dinosaur shown, so you'll want to give them a stable base yourself.

Use breakaway supports to stabilize isolated parts while printing

- Thin strips anchor the part
- Will break away under normal use
- Dramatic increase in print success rate!



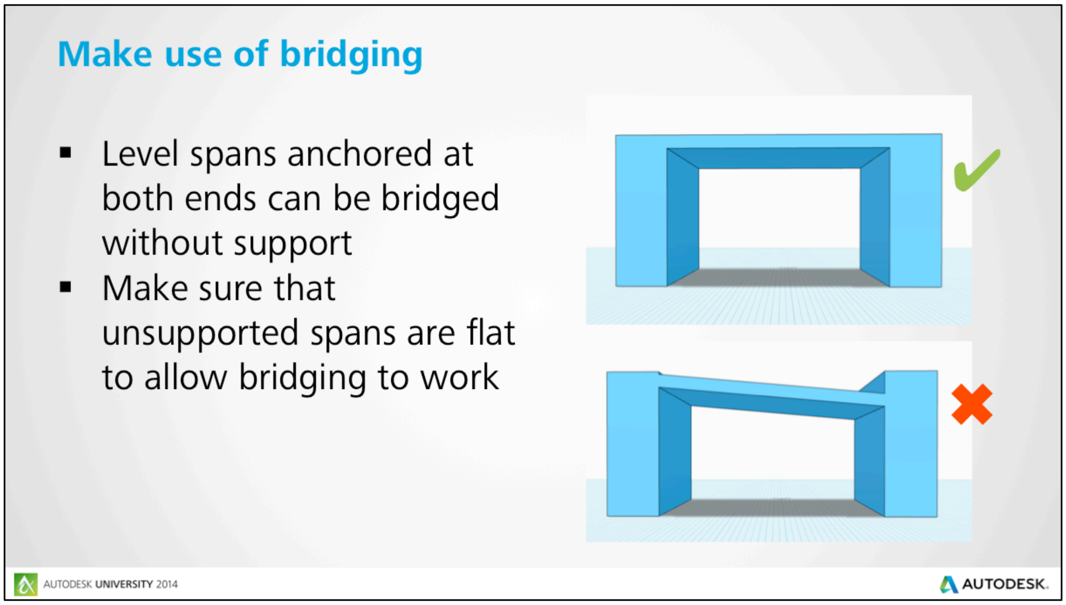
AUTODESK UNIVERSITY 2014 AUTODESK.

Here’s another example of a specialized breakaway support.

The ball on the end of this piece can separate from the print bed, and spoil the print.

Adding a thin bridge between the ball and body stabilizes it while printing, success rate went from 70% to 100%

Finished product…



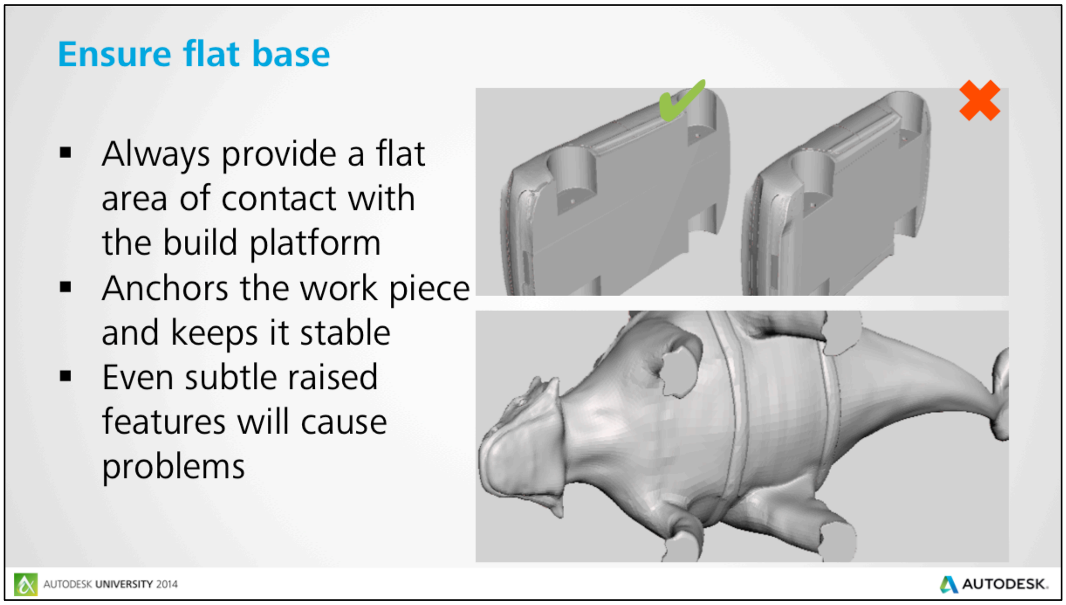
FFF printers are capable of bridging quite large gaps, if both ends are supported. 50mm is certainly possible, and some printers, even greater distances. This is done by reducing the extrusion rate of “bridges” so that the filament is slightly stretched.

However, for this to work, you have to make sure that the spans are completely flat – otherwise you’re dealing with an extreme overhang situation and need to use support.

Advanced bridging

AUTODESK UNIVERSITY 2014 AUTODESK.

This bracelet relies on bridging to create links that are all independent.



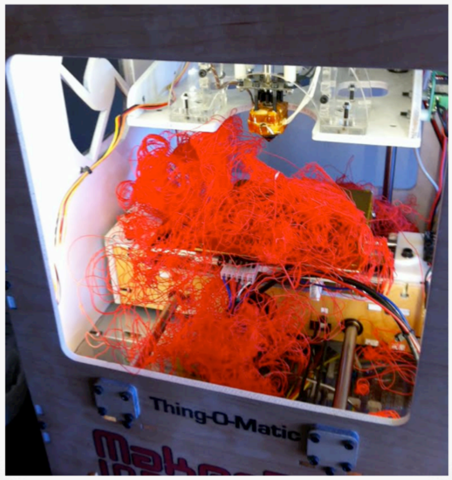
You will want to make sure that there is a consistent flat surface in contact with the build platform. This will keep the piece firmly attached to the build platform.

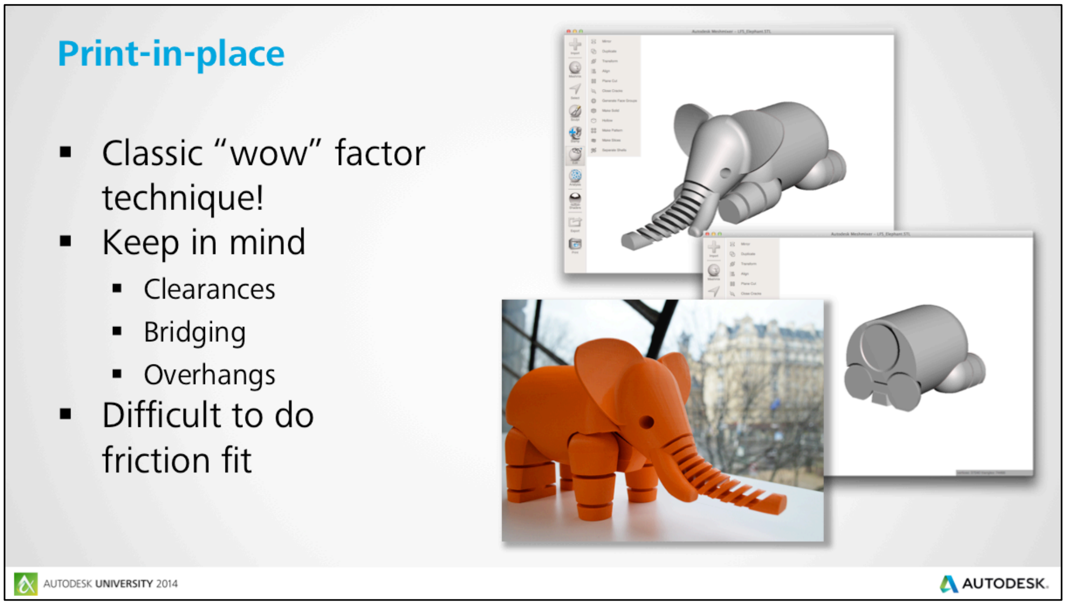
In the example shown, the car on the left has a completely flat body. On the right there is a faintly visible bezel, and the back is an unsupported overhang. If you didn't use support, the car on the right would not stick to the platform very well, and the back would print badly. If you DID use support, you'd have a single layer of support material which would be awkward to remove.

The dinosaur shown has had its feet flattened to make sure that it is well anchored to the build platform. If I only relied on support, it would be a weak attachment.

Rafts can be used in cases like this as well, but they slow down print times and add to post-processing times.

When prints don't stick





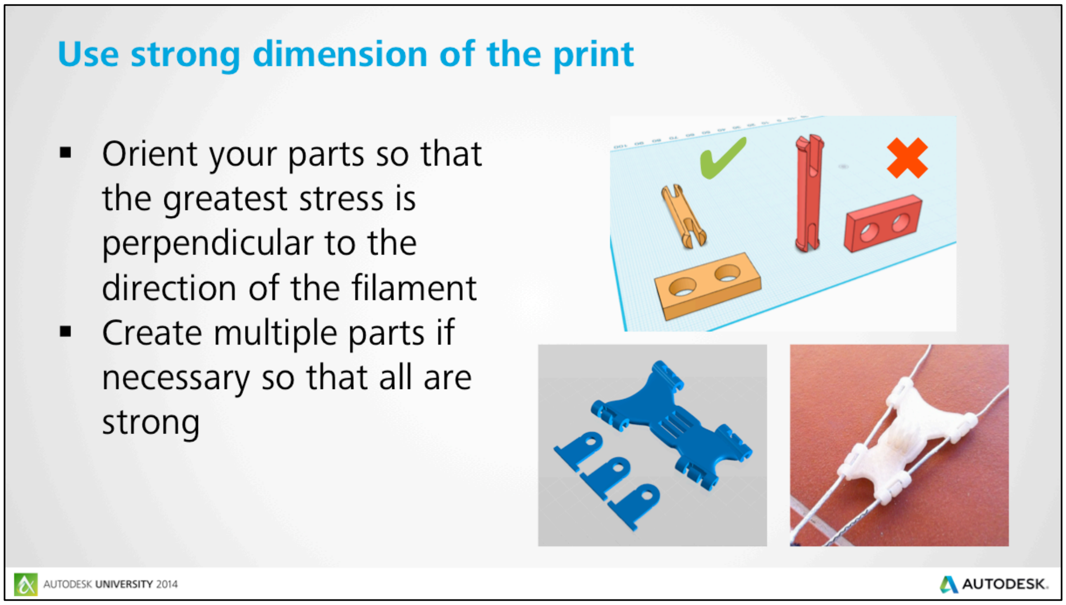
Printing articulated models in one piece is one of the most dramatic demonstrations of the capabilities of 3D printing.

However, for models to work well you have to keep in mind:

- Clearances: allow enough space between moving parts, otherwise they will fuse while printing
- Bridging: use bridging to make shafts that go through other parts
- Overhangs: Use overhangs to create nested parts

Connectors

- Robust and reliable snap connectors let you make complex designs with a minimum of post-processing
- Connectors should take into account material characteristics:
  - Flex
  - Wear



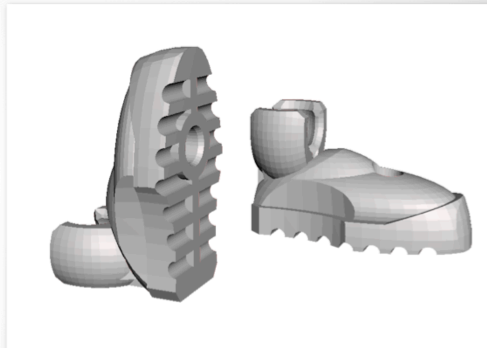
As we reviewed earlier, the adhesion between layers is due to fusing, but is a much weaker bond than that along the length of the filament.

Ideally, we want to orient our parts so that the greatest stress is carried along the length of the filament – the orange parts illustrate this. The red pats would be a lot weaker.

In some cases, it makes sense to split a model into multiple parts so that each makes use of its strongest dimension. The example shown is a mount for a attaching a gopro camera to a stunt kite. The three “prongs” which will be used to attach the camera are made as separate parts, so that the filament “surrounds” the bolt.

Orient parts to optimize strength

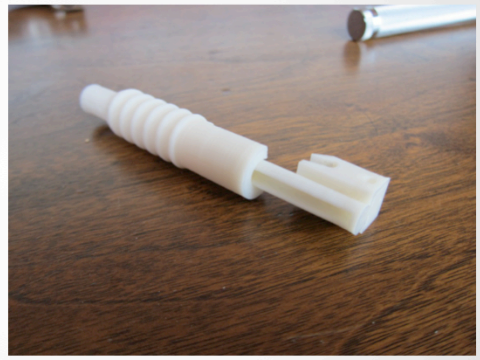
- Socket on the back of the foot will be subject to most stress
- Printing it with the socket flat to the print bed makes it strongest



AUTODESK UNIVERSITY 2014

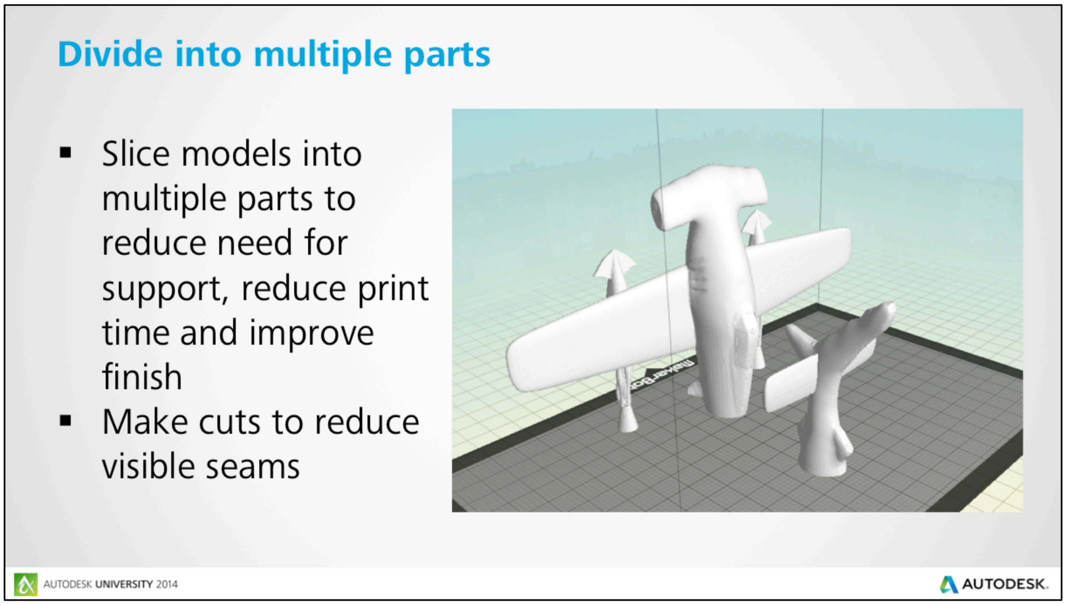
AUTODESK.

Combining parts to maximize strength



AUTODESK UNIVERSITY 2014

AUTODESK.



Judiciously cutting a model into multiple parts for later gluing is a good technique.

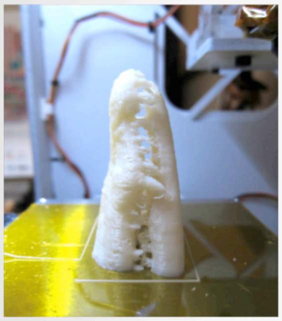
When you have models with details on all sides, it is difficult to find a flat surface to attach to the build platform. Slicing the model can give you this flat surface.

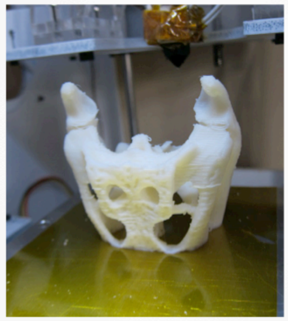
In the example show, multiple techniques are shown – I'm using helper cones to support the pods under wings, there is a flat surface for each part, support will only be used on the trailing edge of each wing, and the final seam will be almost invisible.

If I cut the model along the wing, I could print with even less support, but then I'd have to deal with a much longer and visible seam.

You can use all kinds of adhesive to join your models – cyanoacrylate (Krazy glue), acrylic glue, epoxy, even acetone, depending on your choice of plastic.

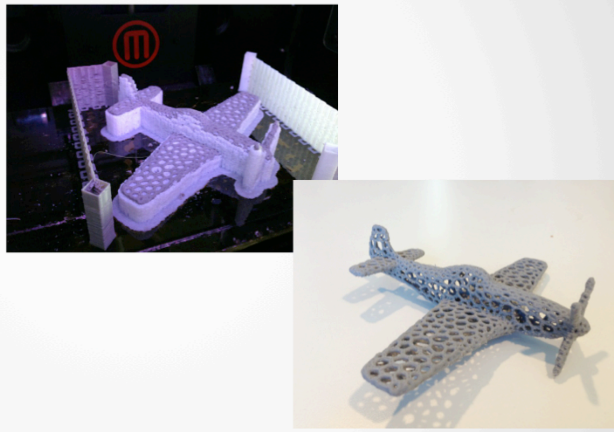
Slice into halves to avoid support





Working with soluble support

- Allows for more complex geometry
- More freedom in part orientation
- Soluble vs single material support requires different optimization



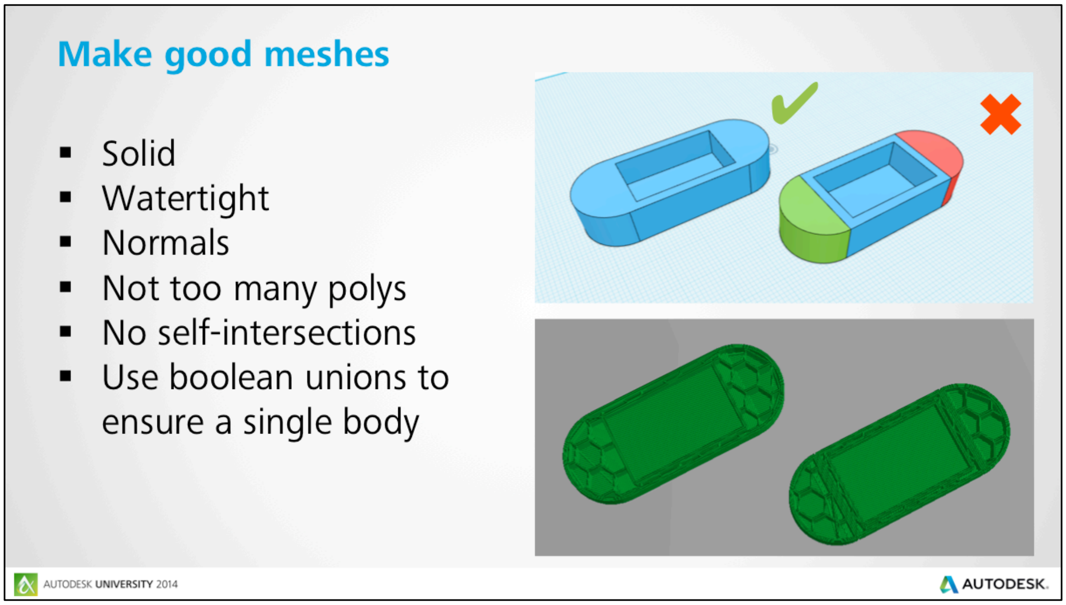
Optimizing soluble support

- Very different from single material support
- Allow for drainage
- Maximize contact area, not minimize
- Defaults are not optimal
- Meshmixer is adding excellent options

Using support material is a great enabler for complex models, however you should use it judiciously.

Orient your parts so as to minimize the amount of scaffolding needed (see the example), and also so that the support attaches to surfaces that are not visible. On the red plate, the support is used to make possible a large overhang on the bottom surface, but it leaves a lot of visible scarring.

Top tip – a lot of this scarring can be removed by using a hot air gun on it for a few seconds.



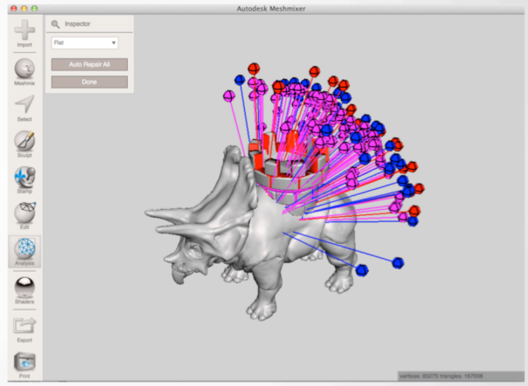
It goes without saying that your meshes should be in good shape. If you are using programs oriented towards CAD or that are solid-aware, it will probably generate a good mesh.

However, it is worth reviewing:

- Solid: avoid surfaces without any thickness
- Watertight: there shouldn't be any gaps or cracks – otherwise it will be treated as a hollow shell
- Normals: make sure that surfaces are correctly oriented facing out – (however a lot of slicers are only counting odd / even surfaces and don't care…)
- No self interesctions: this will throw off the odd / even calculation and lead to strange voids
- Use boolean unions: even if your model looks correct, make sure to use whatever your software allows for to make the model into a single solid – otherwise you'll get strange (and weak) results, as shown in the picture – the model on the right is actually composed of several adjacent solids, and the bond between them when printed

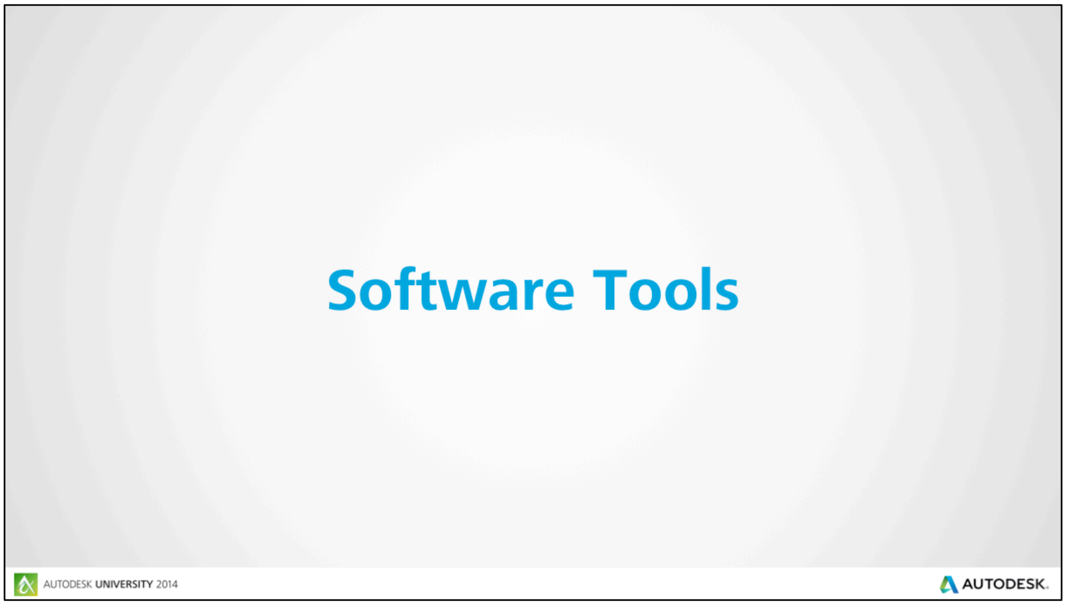
Repair meshes

- Use repair and analysis tools to fix
  - Meshmixer
  - Tinkercad
  - Print Studio



AUTODESK UNIVERSITY 2014 AUTODESK.

Good CAD oriented programs should generate good quality error free meshes – however, if you are using surface modelers, or working with imported meshes, then you should use analysis and repair tools before printing. I’ll go over these in more detail in a bit.

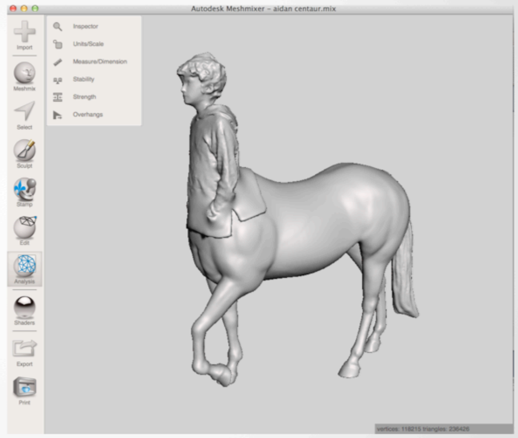


We’ve talked about how these printers work, and explored some design tips and tricks that help us get the most out of them.

Now we’ll talk about some software tools that can help you in the print process.

Meshmixer

- Superb tool for working with meshes
- Sculpt, remix, edit and paint
- Great new tools for pre-print (Make Solid, Support, Patterns)
- Identify overhangs and generate support
- Powerful pre-print analysis tools
- Integrates 3D print utility

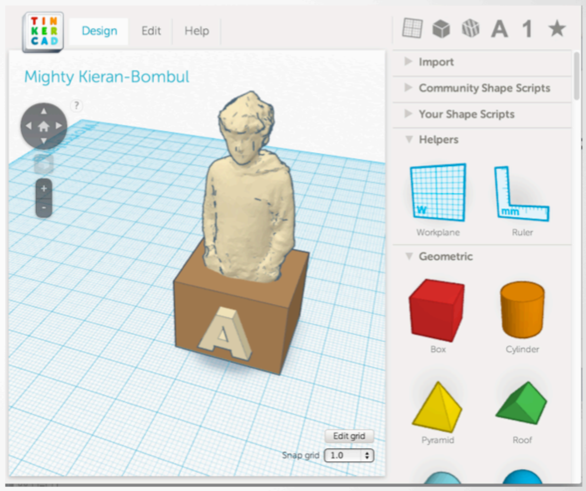


www.123dapp.com/meshmixer

Tinkercad

- Excellent online tool for working with simple geometry
- Can import, modify and export STL files
- Superb automatic mesh repair

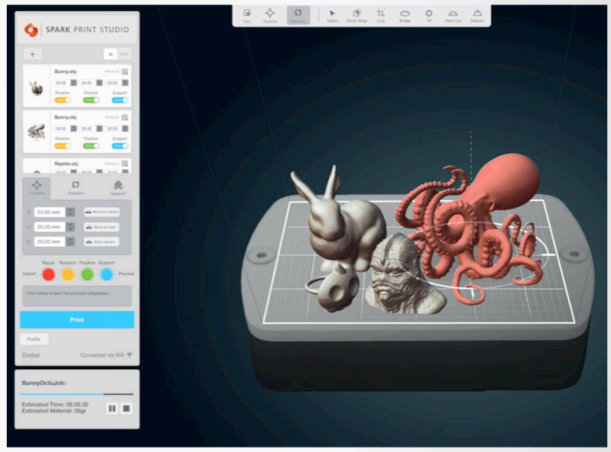
tinkercad.com



Print Studio

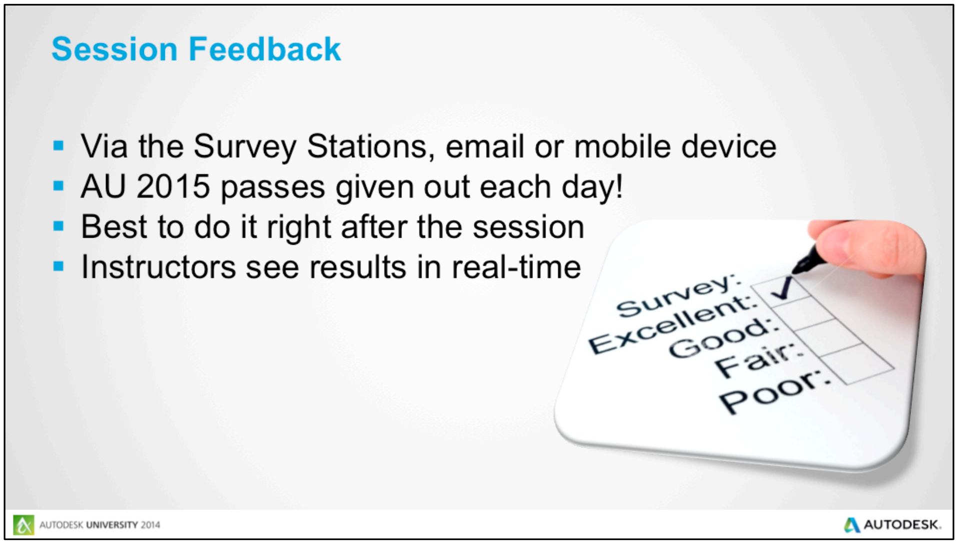
- Autodesk's integrated tool for 3D printing
- Advanced support
- Healing and repair
- Layout
- And more…

spark.autodesk.com



Conclusions

- Consumer level 3D printers can make strong, practical parts – in fact this is the main use case
- In order to get the most benefit, it makes sense to design specifically for the characteristics of these printers
- Fortunately, the constraints and design rules are simple
- Tools to get good results are becoming better and more accessible



We really value your feedback – after a class is completed you will receive an email with a link to complete a class survey…

You can respond to that email on your computer, mobile device or via the Survey stations that are all over the place here at AU.
The instructors compete for the prestigious Instructor Awards –
 and we also use your input on classes to determine the classes for next year's AU.
And each day we will give away two free passes to next year's AU…

You heard it from Carl on main stage, Autodesk is now providing free* access to Autodesk design software and apps to students and academic institutions – worldwide. This includes full versions of Autodesk software, same as the professionals.

Autodesk believes the next generation will create a better world and wants to inspire a love of design and equip in students of all ages to become engineers, architects, and digital artists. That’s why Autodesk provides free* access to professional Autodesk design software and apps to students and academic institutions.

Questions?

AUTODESK UNIVERSITY 2014

AUTODESK.