SolidWorks & Tinkerine 3D Printing Tutorial

ELEC391

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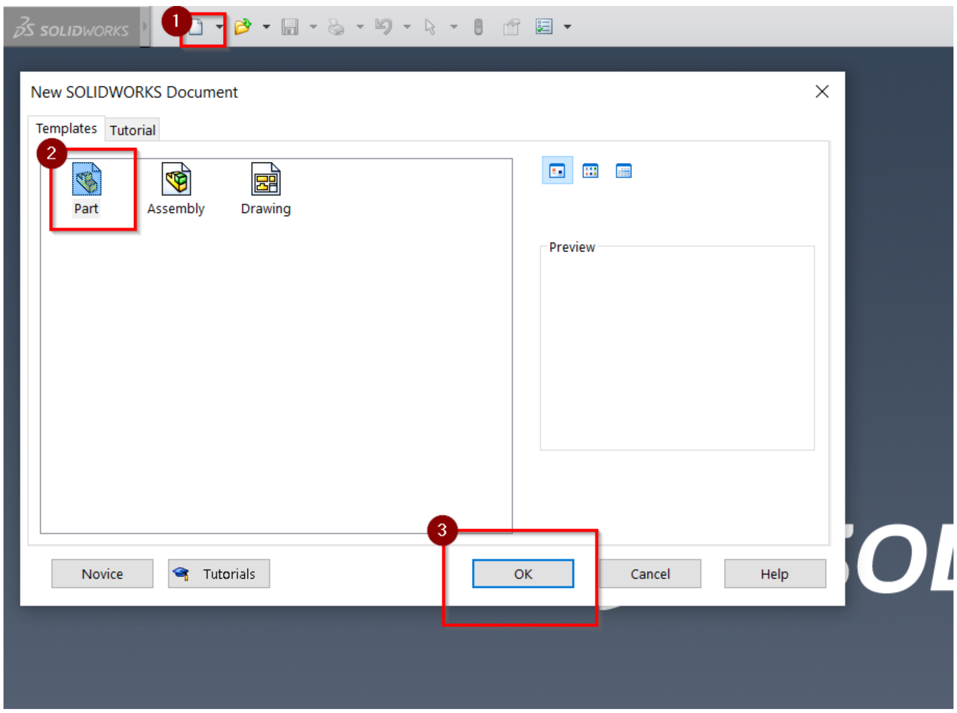
Installing SolidWorks and Initial Setup

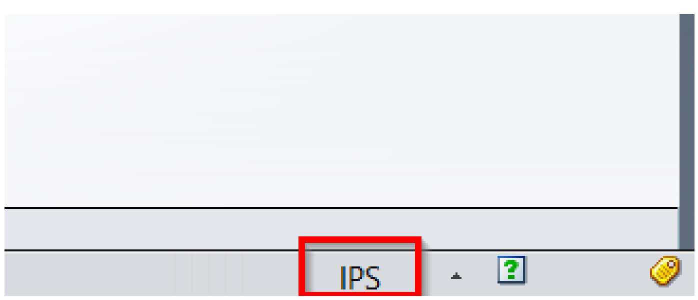
A SolidWorks licence is available to all ECE students free of charge. To obtain the software, follow the instructions below or go to http://eng-services.ece.ubc.ca/software/software-master-list/classroom-use-software/. You will need your ECE login credentials to obtain the SEK and serial number. Once SolidWorks is installed, proceed to the initial setup phase.

For the purposes of following this tutorial it is important to make sure SolidWorks is set up in metric units MMGS (mm, g, s).

1. Open up SolidWorks and start a new part. Be sure to select Part and not select Assembly or Drawing.

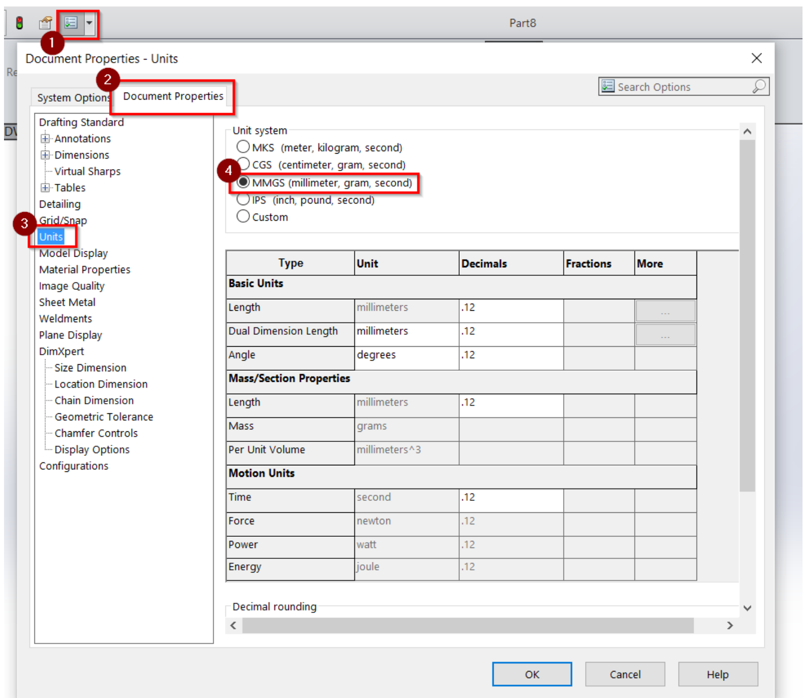


2. You may notice at this point that the units in the bottom right corner shows "IPS". This means the unit system is imperial: inches, pounds, and seconds. SolidWorks will usually be set up by default in this unit system.

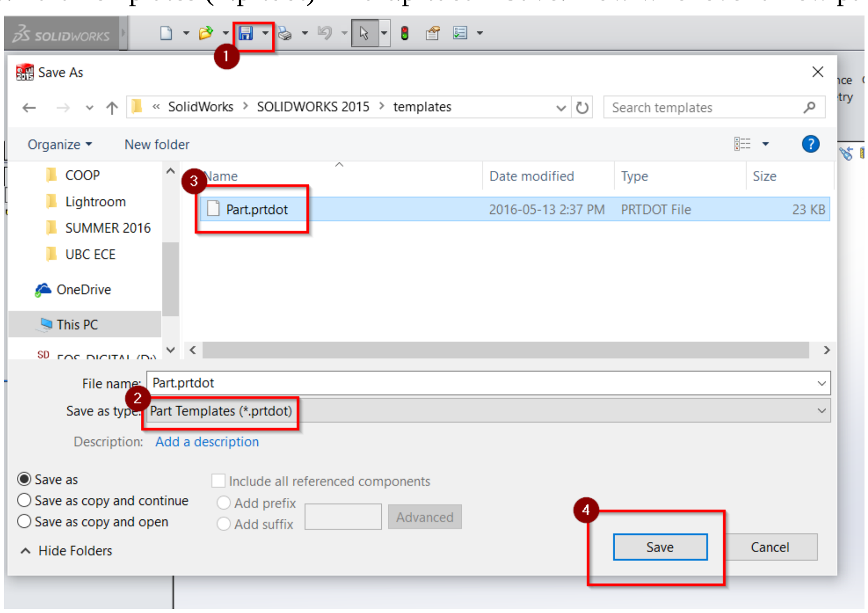


3. To change the default unit system, navigate to the Units menu in the options:

    i. First, click on the options button in the top toolbar.
    ii. Next, switch to the Document Properties tab
    iii. Go to the Units menu
    iv. Check the “MMGS (millimeter, gram, second)” option. Hit OK to finalize.



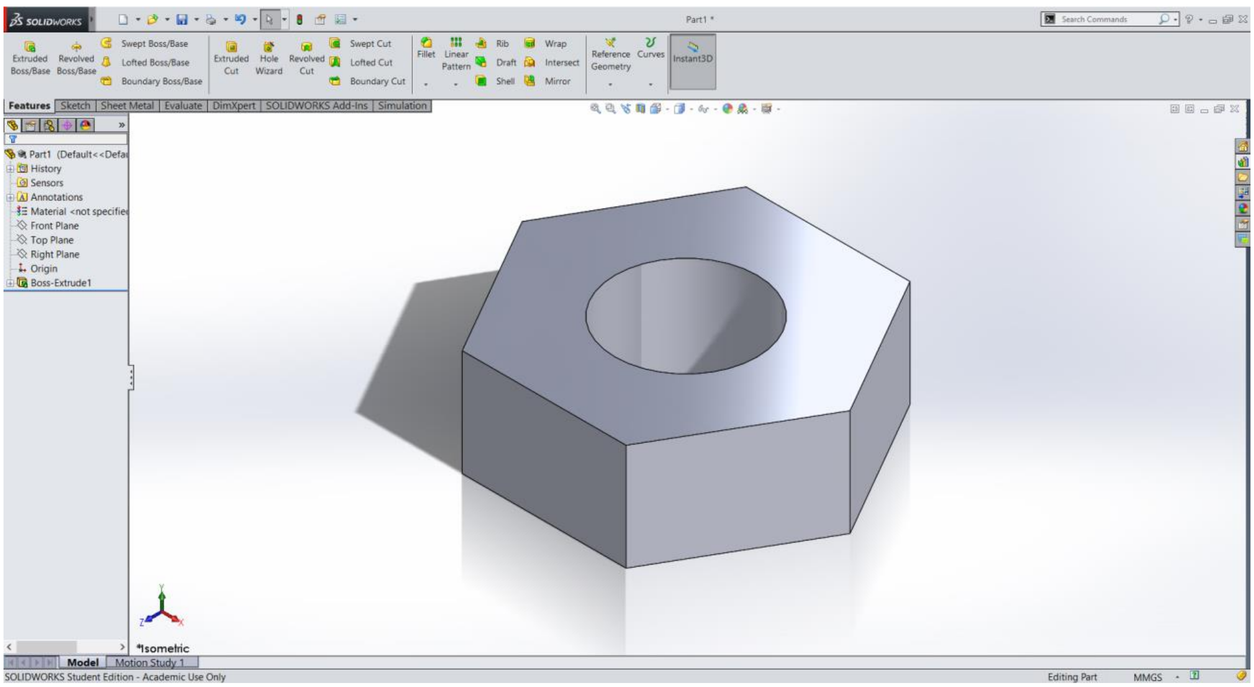
4. Next, save as a template, replacing the default template with the new one you just created: Save As > Save as type: Part Templates (*.prtdot) > Part.prtdot > Save. Now whenever a new part is started, the unit system will automatically be in metric units.

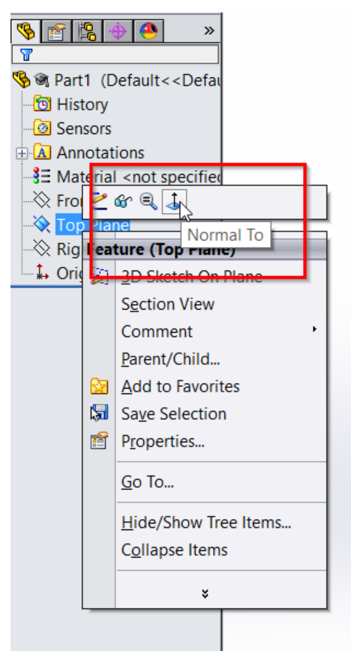


Exercise 1: Basic Hex Nut

Features Learned: Sketch (Polygon, Circle), Smart Dimension, Extruded Boss/Base

In this first exercise we will design a simple hexagonal unthreaded nut as seen in the figure below. This exercise will build the foundations of how to design 3D parts in SolidWorks and introduce several fundamental features such as Sketching, Dimensioning and Extruding. Let's begin.

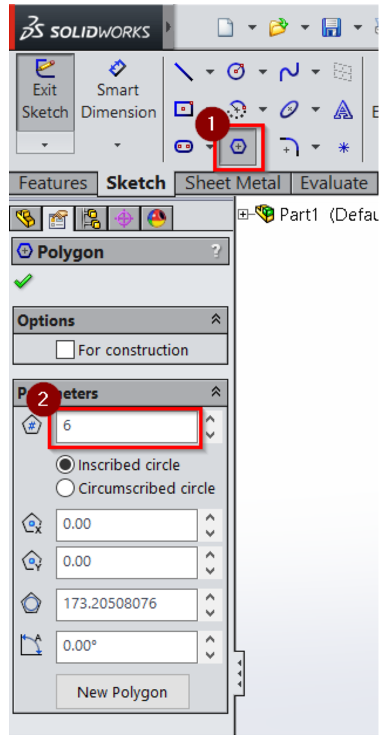




1. First, start a new part by going to New > Part > Ok. Since we have already set up SolidWorks to use MMGS as default units, all our units should be in millimeters, grams and seconds. You should now be greeted with the main drafting interface of SolidWorks.

2. On the left hand side you will see the FeatureManager Design Tree. Right click on "Top Plane" and click the "Normal To" option. Now we are facing the Top Plane head-on.

3. While the Top Plane is selected, switch to the Sketch tab at the top of the interface and select the "Sketch" tool in the top left corner. We are now creating a sketch on the Top Plane. Anything we sketch will appear on the Top Plane.



First we want to sketch the basic hexagonal outline of the nut.

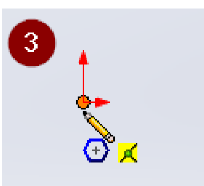
i. Click the "Polygon" tool in the Sketch tab. A list of parameters should appear in the left hand panel.

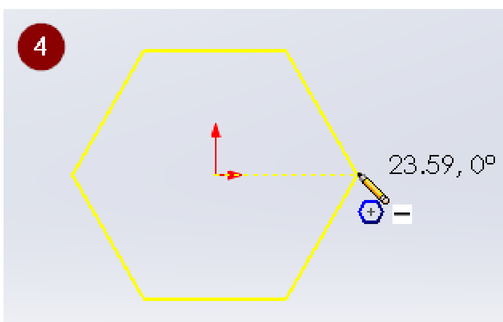
ii. Set the number of sides to 6 (hex = 6). Other parameters such as center position, angle, etc. are also modifiable, but we will not adjust these.

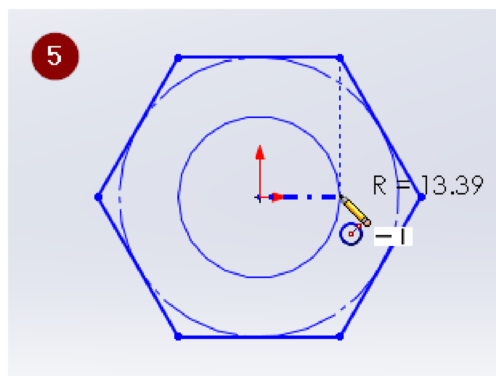
iii. Next, hover over the origin (the base of the two red arrows) until the cursor snaps onto the dot in the middle. Left click once to start the sketch.

iv. Left click again at some point to the right of the first point. Press enter to finalize the sketch.

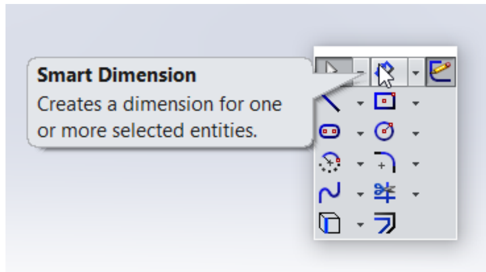
v. Now, select the "Circle" tool in the Sketch tab and sketch a circle starting at the origin, similar to how we sketched the hexagon. Again, do not worry about the exact size of this circle, but ensure that it is centered at the origin and is smaller than the hexagon. Press enter to confirm the circle sketch.







4. Now we will assign dimensions to our outline. Make sure we are still in sketch mode by looking at the top left corner and seeing that the Sketch button is depressed and now reads “Exit Sketch”. Press the “S” key on the keyboard. In the small window, click the “Smart Dimension” tool.

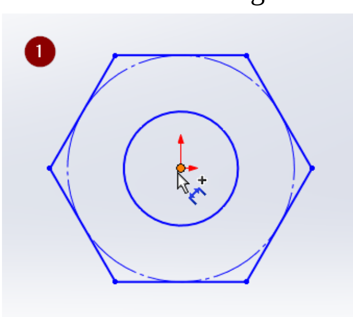


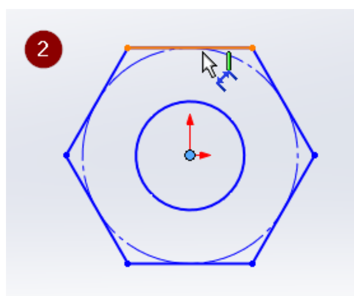
Now let’s set the center-to-edge dimensions of the hexagon,

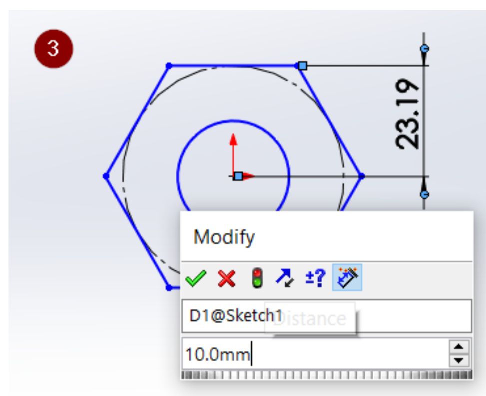
i. Left click the center

ii. Then left click the top side (be sure to select a side edge and not one of the points between two edges).

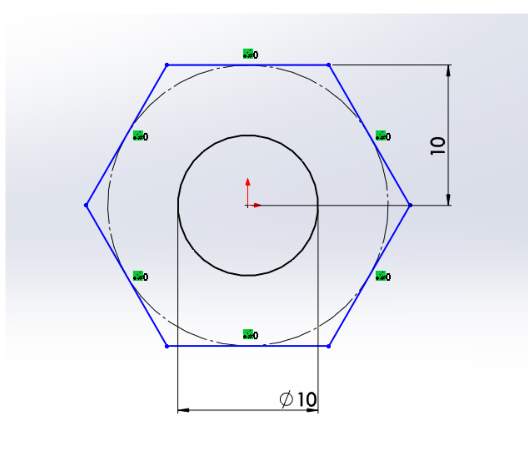
iii. A dimension measurement should pop up. Left click again anywhere on the screen and a small window will appear. Enter 10mm into this field and hit enter. The hexagon is now assigned a dimension of 10mm (center to edge). Double click the dimension label to change the assigned dimension at any time.





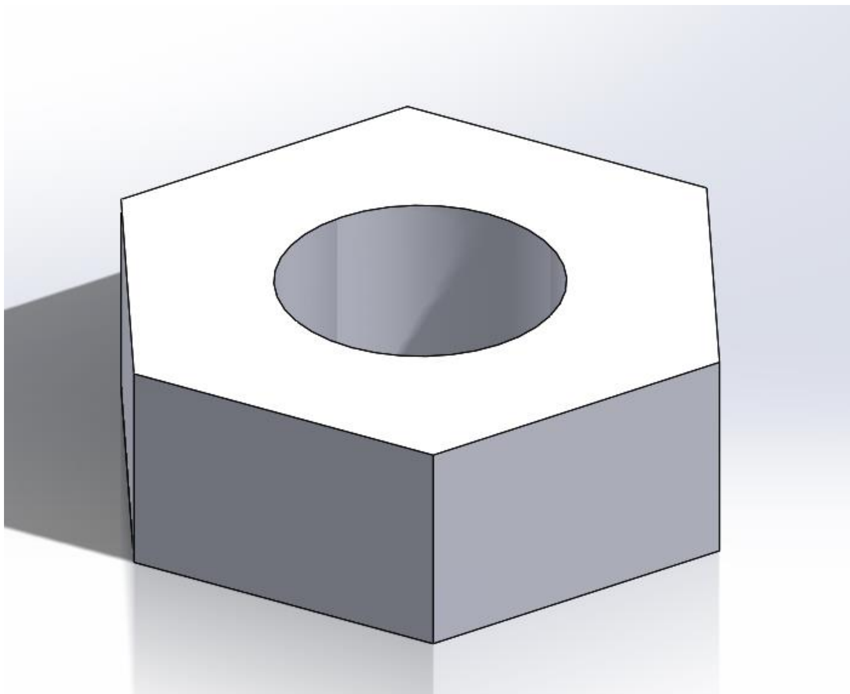


Follow the same procedure to set the diameter of the circle to 10mm. Select Exit Sketch when finished. The sketch should now appear as follows:



5. Now that our sketch is assigned dimensions and finalized, we are ready to bring this shape into the 3rd dimension. Select our sketch by clicking “Sketch1” in the left panel. The sketch should turn light blue as it is selected.

   Next, switch to the Features tab at the top and click the “Extruded Boss/Base” tool. Immediately you should see the shape gain depth. Using the parameters on the left panel, give the nut a depth of 7.5mm. Hit enter to confirm the extrusion. The shape should now appear as follows:

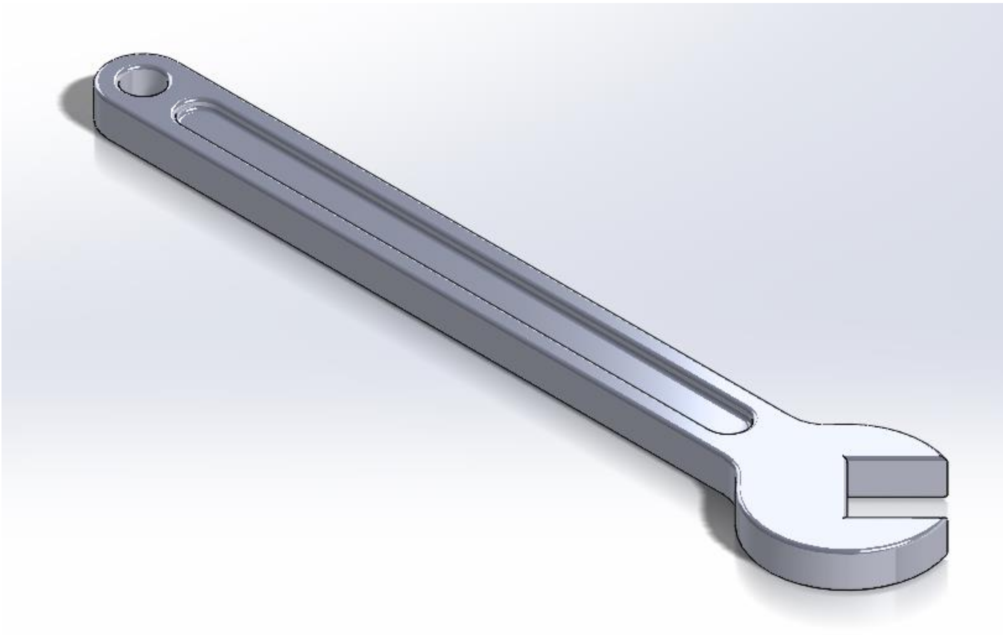


6. At this point we are done drafting our first simple part. Hold down the middle mouse button and move the mouse around to view different angles of the part. Hold CTRL+middle mouse button and move around to pan. Scrolling the middle mouse button will zoom in and out. Save this part as a *.prt;*.sldprt file. We will be using this part in later tutorials.

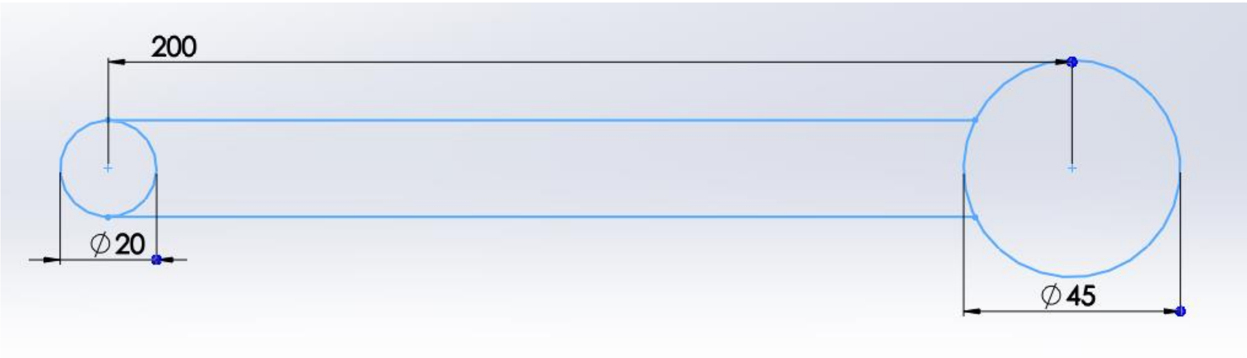
Exercise 2: Simple Wrench

Features Learned: Extruded Cut, Fillet, Add Relations

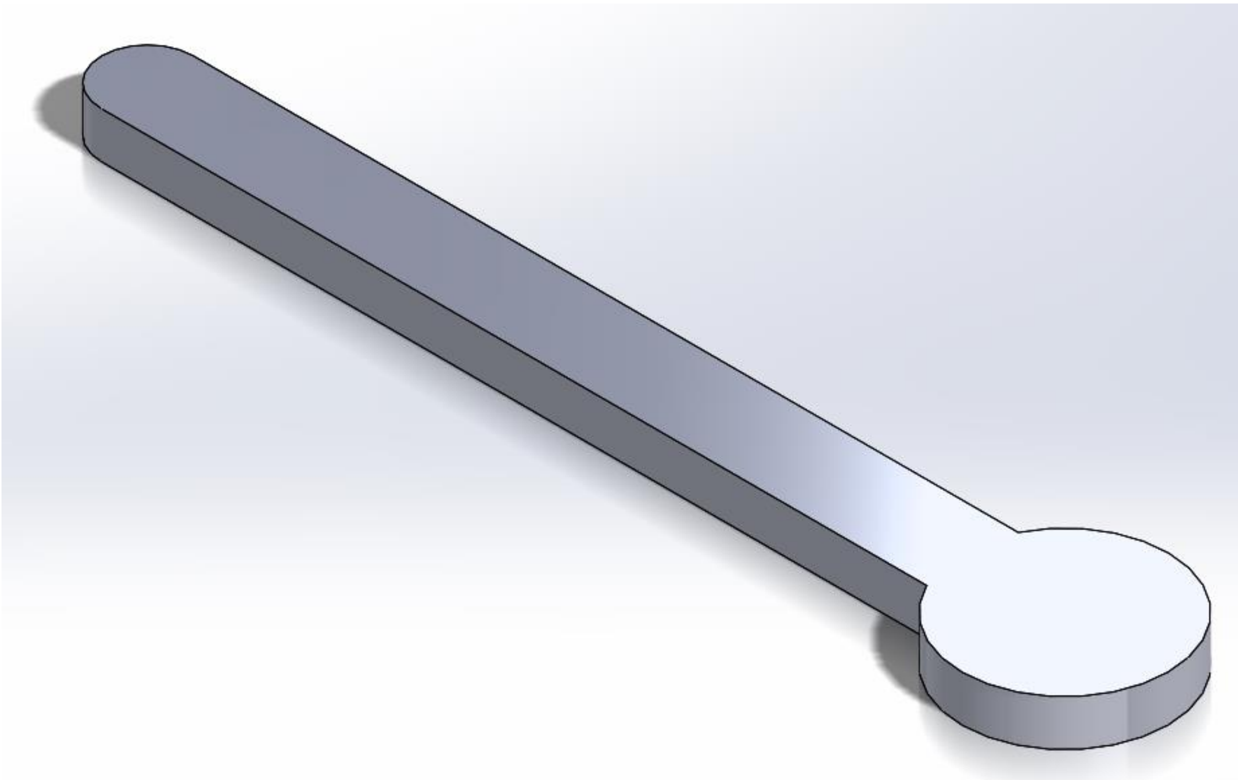
In this exercise we will be building on the basic tools we learned last time. In addition to the basic features taught in the last exercise, we will be practicing some new tools such as extruded cuts, fillets, relations, trim entities, etc. to design a simple wrench as seen below.



1. First, start a new part and sketch the shapes below on the top plane according to the given dimensions. NOTE: the lines are tangent to the top and bottom of the smaller circle.



2. Make sure this sketch is selected and use the extrude tool to extrude this shape by 10mm. We now have our basic wrench shape. From this part we will cut away pieces and round edges to obtain our final product. NOTE: you will have to select all 3 contours when extruding this shape, otherwise only one region of the wrench will be extruded.



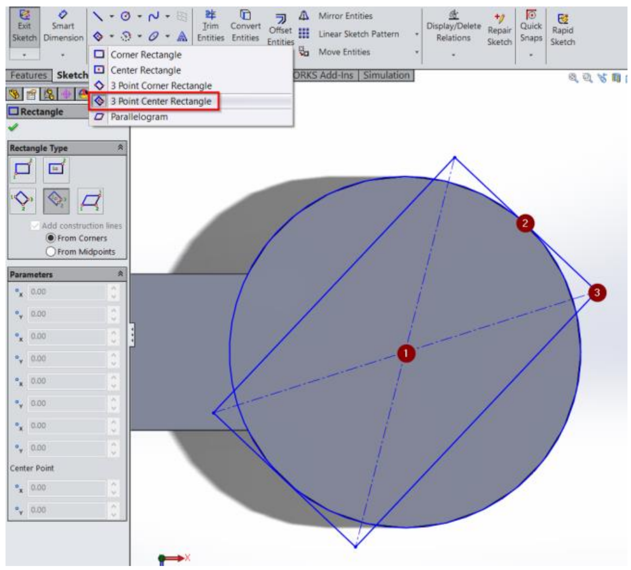
3. Now make a sketch on the top surface of the wrench, instead of making a sketch on the top plane, as we have done so far. Simply right click on the top surface of the wrench and click the sketch tool. Sketch the shapes as shown below, starting with the circle on the left.

To sketch the shape on the right, simply start with a circle that is identical in shape and size to the original circle. Next, select the 3 Point Center Rectangle tool (click the small triangle next to the rectangle tool to access more options).

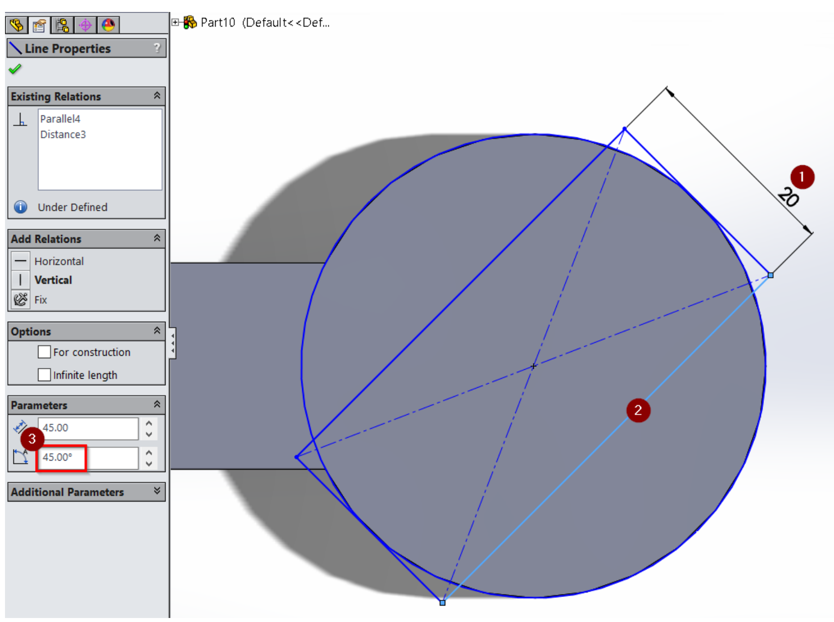
i. First click the center of the circle (if you are having difficulty snapping onto the center of the circle, first place your cursor over the outer arc, and the center should appear)

ii. Then click the edge of the circle at about 45° North East

iii. Extend the rectangle until it takes up most of the wrench head as shown below.

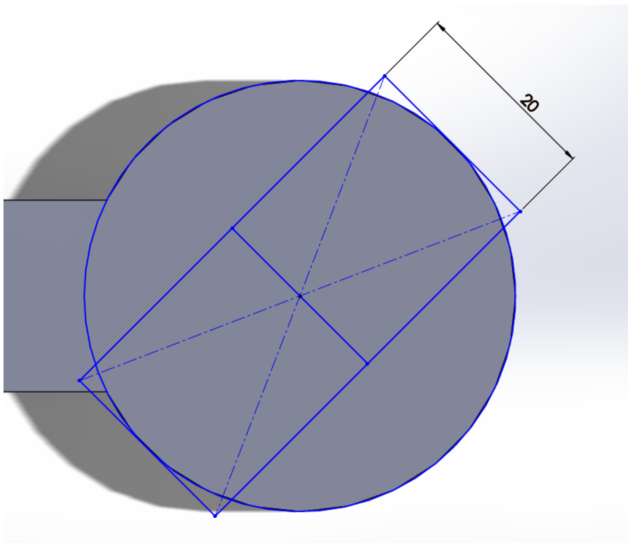


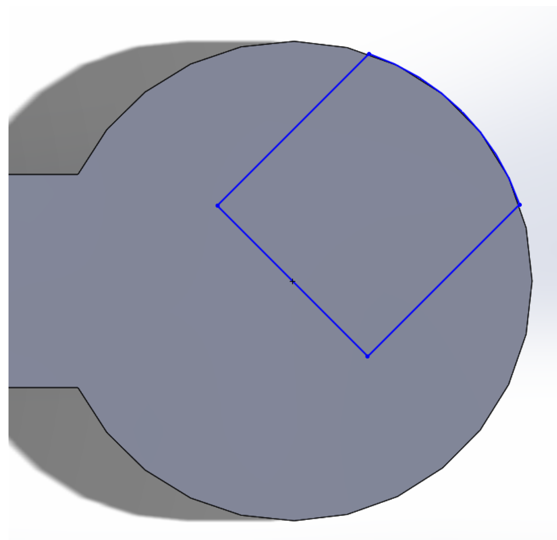
Now set the width of the rectangle to 20mm (to fit our hex nut from the previous example). Afterwards, set the angle of the rectangle to 45° by selecting one of the longer sides and choosing 45° in the parameters panel on the right.



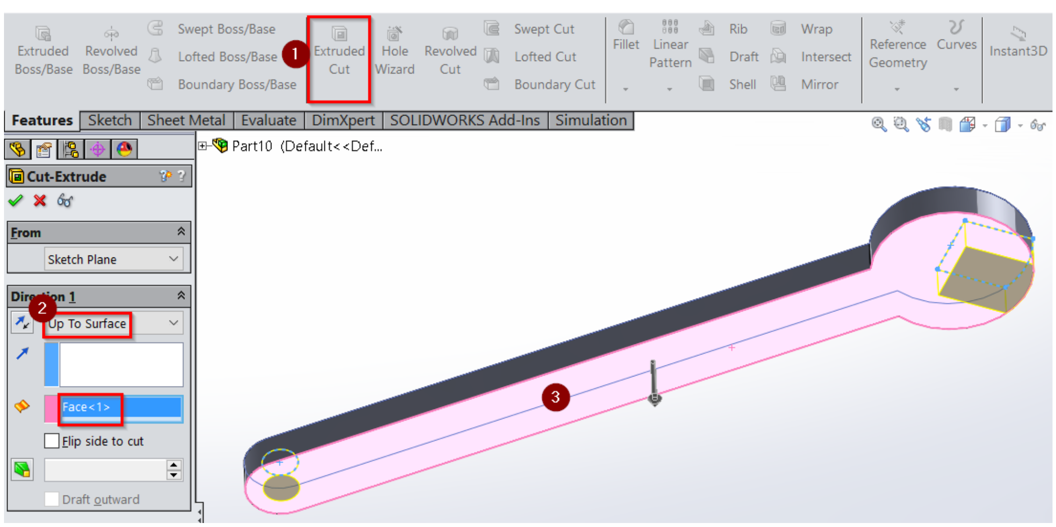
At this point the center of the rectangle may have shifted. If this is the case, simply select both the center of the rectangle and the center of the circle (CTRL + left click) and selecting Midpoint in the Add Relations panel on the left side.

Next, add a line down the middle of the rectangle. Then trim all unnecessary lines (using the Trim Entities tool) until you are left with the shape below. To use the Trim Entities tool, simple select the tool, then left click and drag across any lines or entities to remove them.

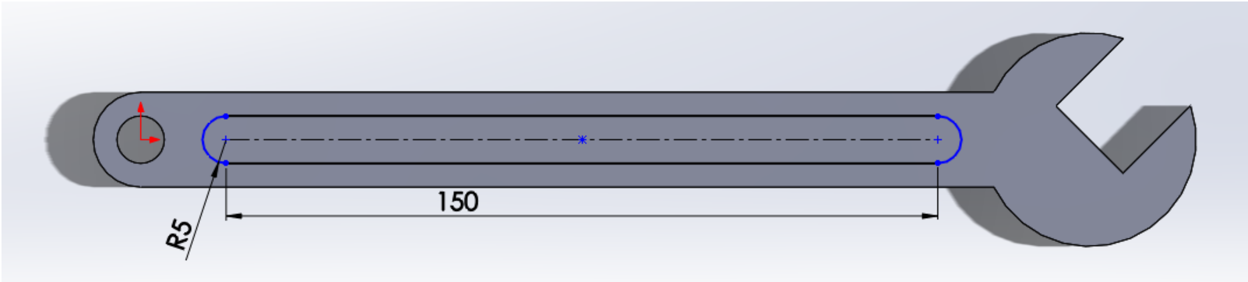




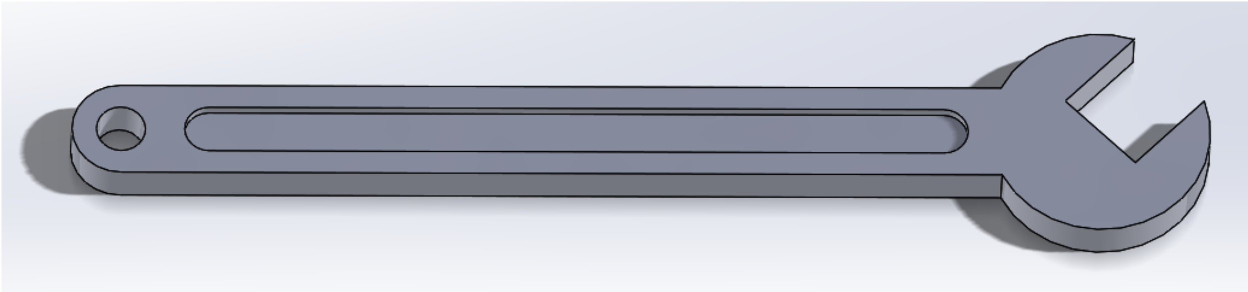
4. Once we have these two shapes we will use the Extruded Cut tool to cut these shapes out of the wrench. With the sketch selected, simply click the Extruded Cut tool in the features panel, select Up To Surface as the End Condition, and select the bottom surface as the Face/Plane. This will cut these two shapes up to the bottom surface. It is also possible to use Blind as the End Condition and cut down 10mm, but if we decide to increase the depth of the wrench later on, our Extruded Cut will no longer pass through the entire wrench. By using Up To Surface we ensure that the cut will pass through the entire part, regardless of changes to the part's depth.



5. Next we will add the grip indent. Make a separate sketch on the top surface of the wrench, similarly to last time. Use the Straight Slot tool to sketch the following shape on the top surface.



Extrude Cut this sketch down, but not all the way through. Cut 2.5mm into the part. At this point our wrench is almost done, the only thing left to do is fillet the edges.

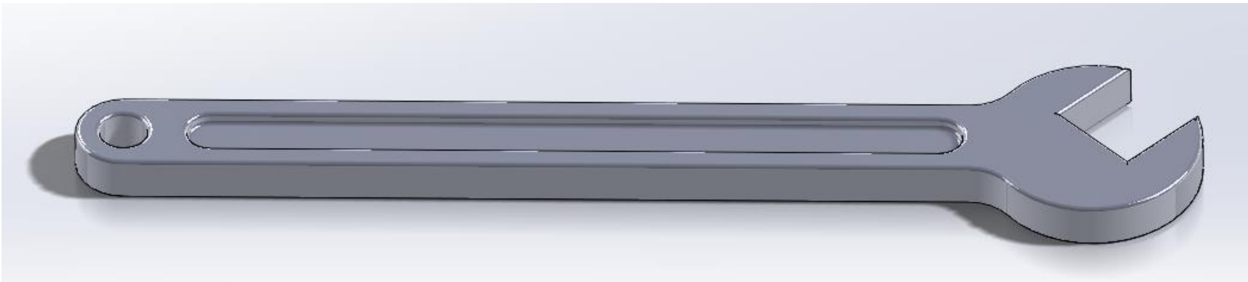


6. Fillets are crucial when considering the mechanical properties of a part. Fillets reduce stress concentrations, eliminate dangerous sharp edges, and make the part more ergonomic.

   We will first add a fillet at the edges where the wrench head and shaft meet. This area is of particular interest since the torque applied by the wrench will result in a concentration of stresses on this edge. To ensure the wrench does not fail, we will add 10mm fillets to these two edges. Select the fillet tool, click both edges where the head meets the shaft, and select 10mm in the fillet parameters panel.

   Next add 1mm fillets to the top surface, the bottom surface, and the bottom surface of the grip indent. NOTE: the fillet radius must be constrained by the geometric properties of the surface. Since our grip is 2.5mm deep, we cannot have fillets greater than 1.25mm.

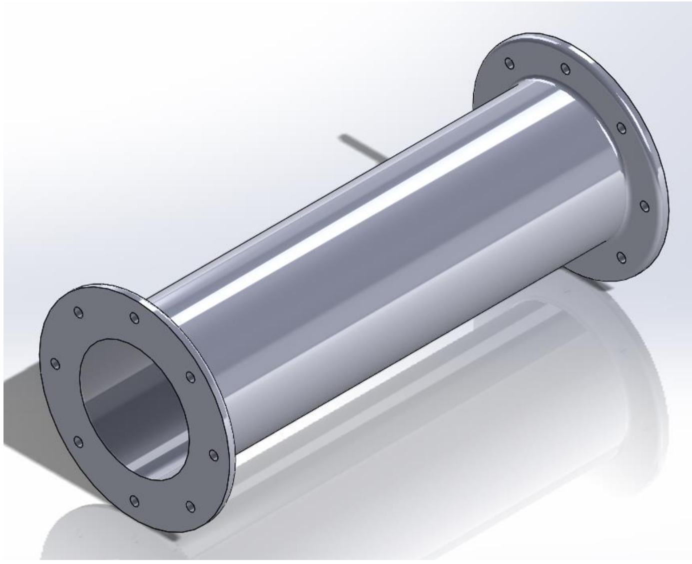
   The result is a nicely rounded wrench with supporting fillets at the base of the head.



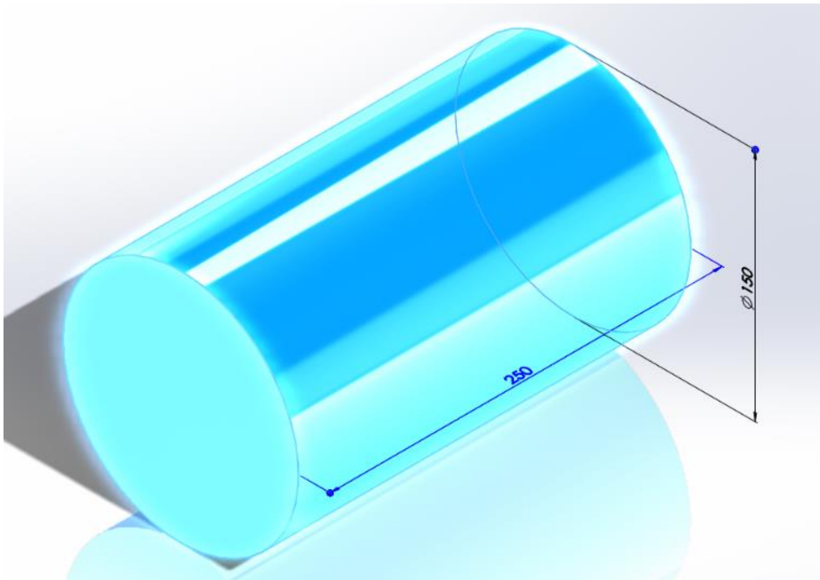
Exercise 3: Pipe Section

Features Learned: Mirror, Circular Sketch Pattern, Shell

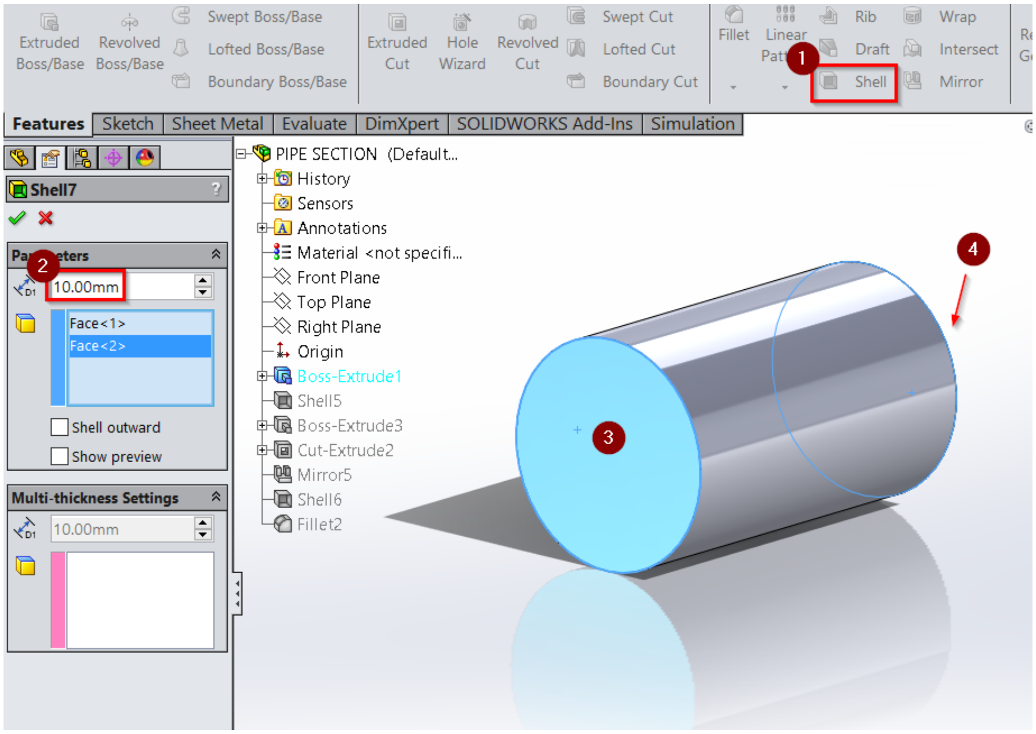
In this exercise we will be designing a small section of a pipe as shown below. At this point you should feel very comfortable using SolidWorks to draft basic mechanical parts. In this lesson we will introduce some new features but overall this exercise should not take long to complete.



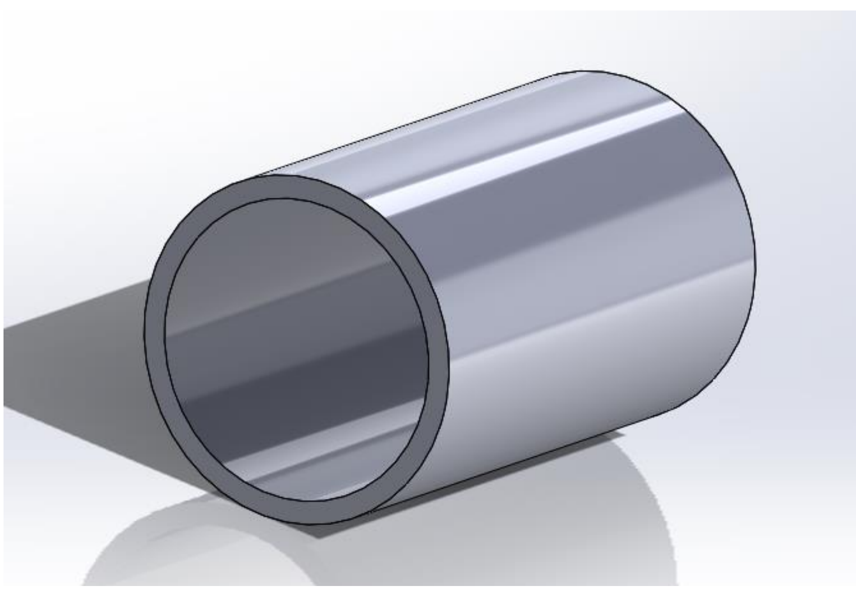
1. First sketch and extrude the shape below. Take note of the dimensions and the extrusion length.



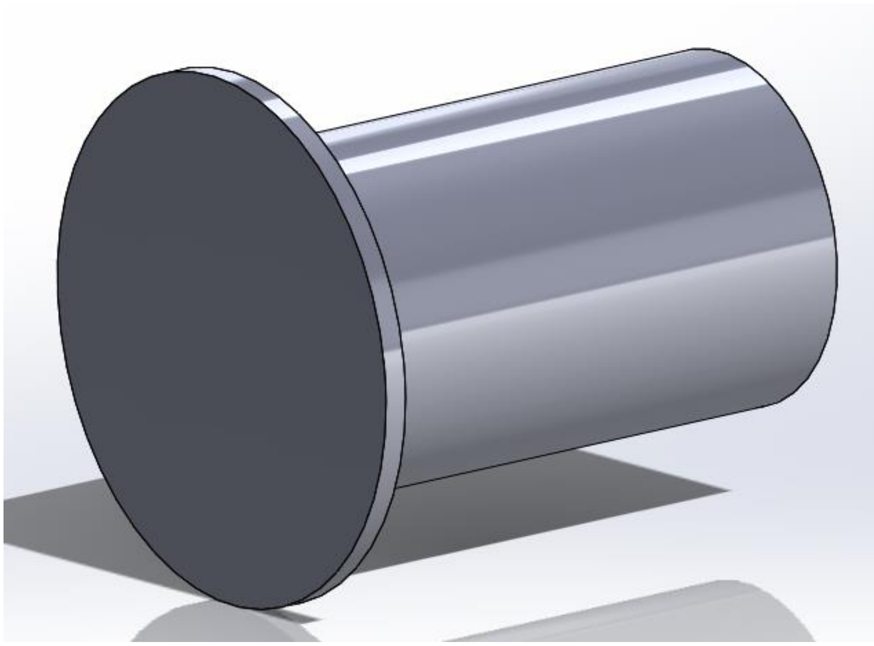
2. Next, use the Shell tool to hollow out the inside of this cylinder. The Shell tool is found in the features tab. Select the Shell tool, click both the circular faces for the Faces to Remove and set the Thickness to 10mm.



The resulting part should look as follows:

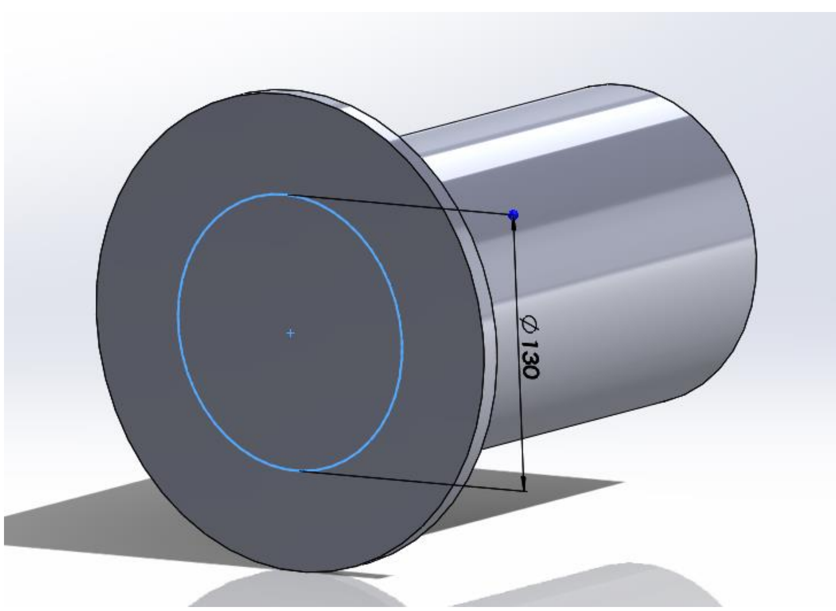


3. Once the cylinder has been shelled, add a flared base to the pipe section for fastening onto other sections. Do this by simply making a circular sketch on one face of the cylinder, and extruding the sketch by 10mm. The base should be 225mm in diameter.

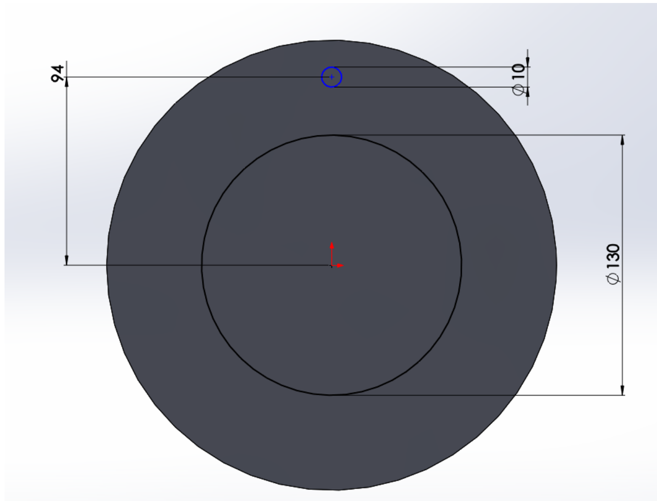


4. Next, we will cut holes in this base for both the bolts and for the material passing through the pipe.

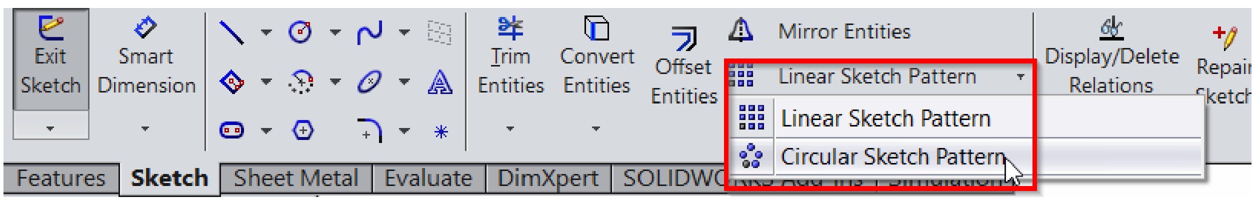
   First, simply sketch a circle on the surface of this base. This circle should be the same as the inner diameter of this pipe, so 130mm (150mm – 10mm shell x 2 = 130mm)



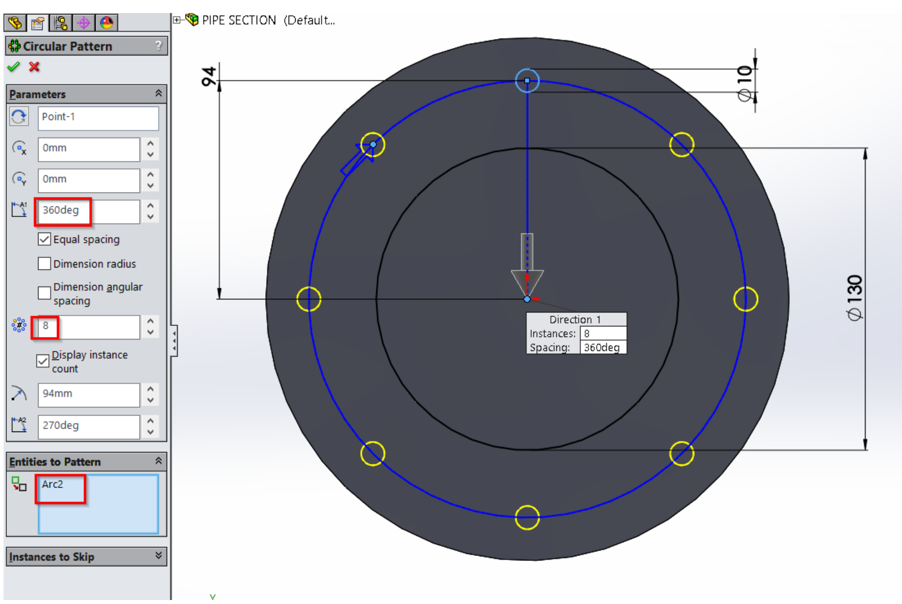
Next, we will use the Circular Sketch Pattern to sketch a series of circles around the outside of the base. First add a circle at the top of the base, 10mm in diameter and 94mm from the center.



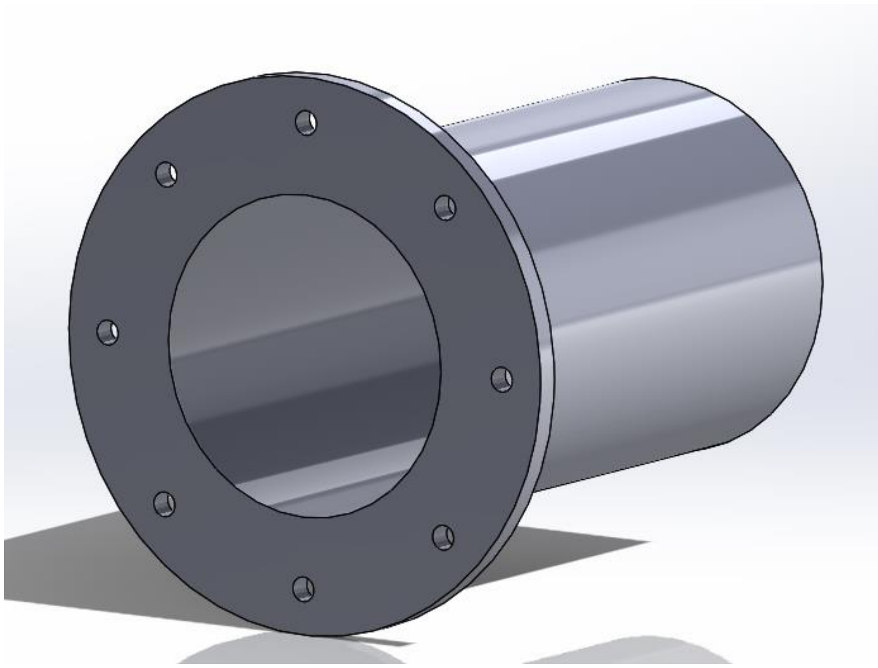
Next, while still in sketch mode, select the Circular Sketch Pattern tool (click the small arrow next to linear sketch pattern).



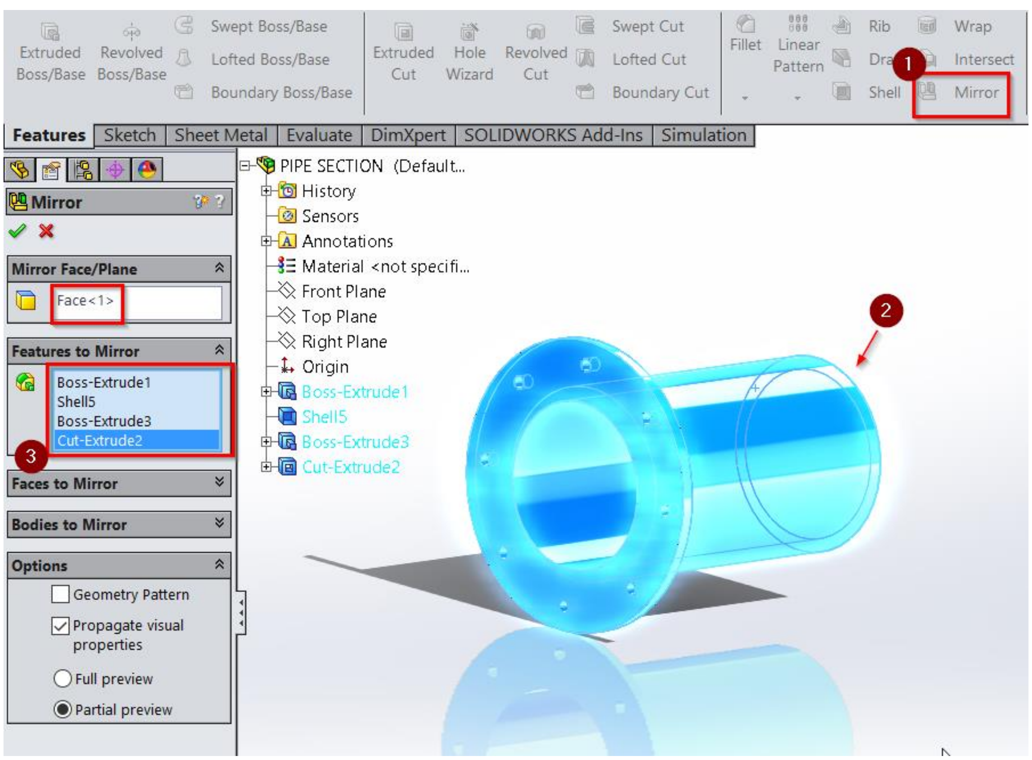
Ensure that the angle spacing is at 360°, and set the Number of Instances to 8. In the Entities to Pattern field, select the 10mm circle.



Now extrude cut this sketch through the entire depth of the base. The resulting part should look as follows.

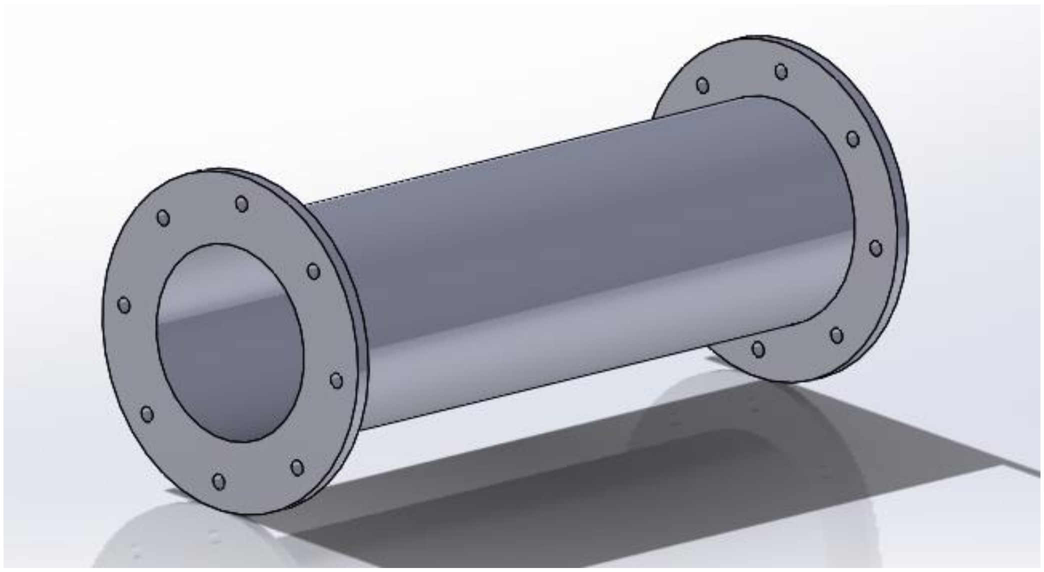


5. The next step is to mirror this part so we have a full length section of pipe with bases at either end. To do this we will use the Mirror tool in the features tab. Select the mirror tool, then select the face without the base as the Mirror Face/Plane. In the drop-down menu on the left side, select all 4 features as the Features to Mirror.

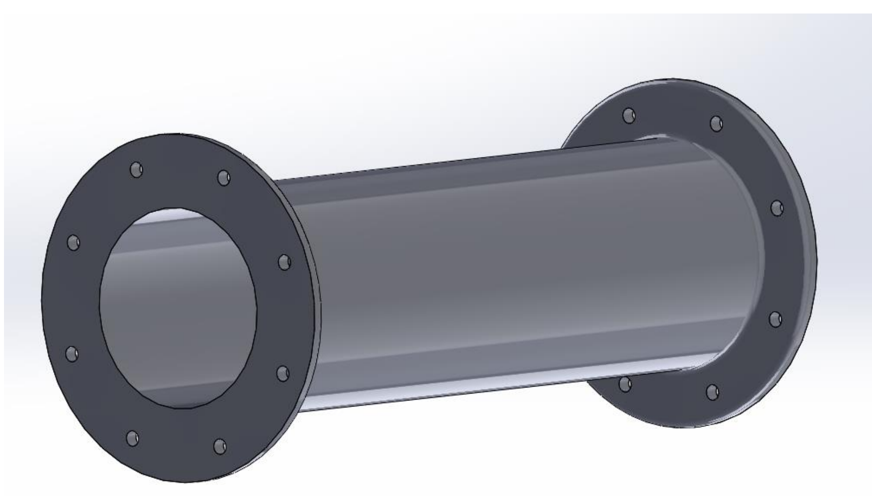


Once the steps are followed we should have a full length pipe section with bases at both ends. Note how the mirrored half does not have the 10mm shell. This is because we applied the shell to one half of the section before we applied the mirror. If we applied the mirror first, and then applied the shell to the entire section afterwards, the whole section would be hollow. This is a common consideration when dealing with SolidWorks; sometimes features do not carry over when other features are applied afterwards.

In this instance we can simply shell the mirrored section again to achieve the hollow inside.



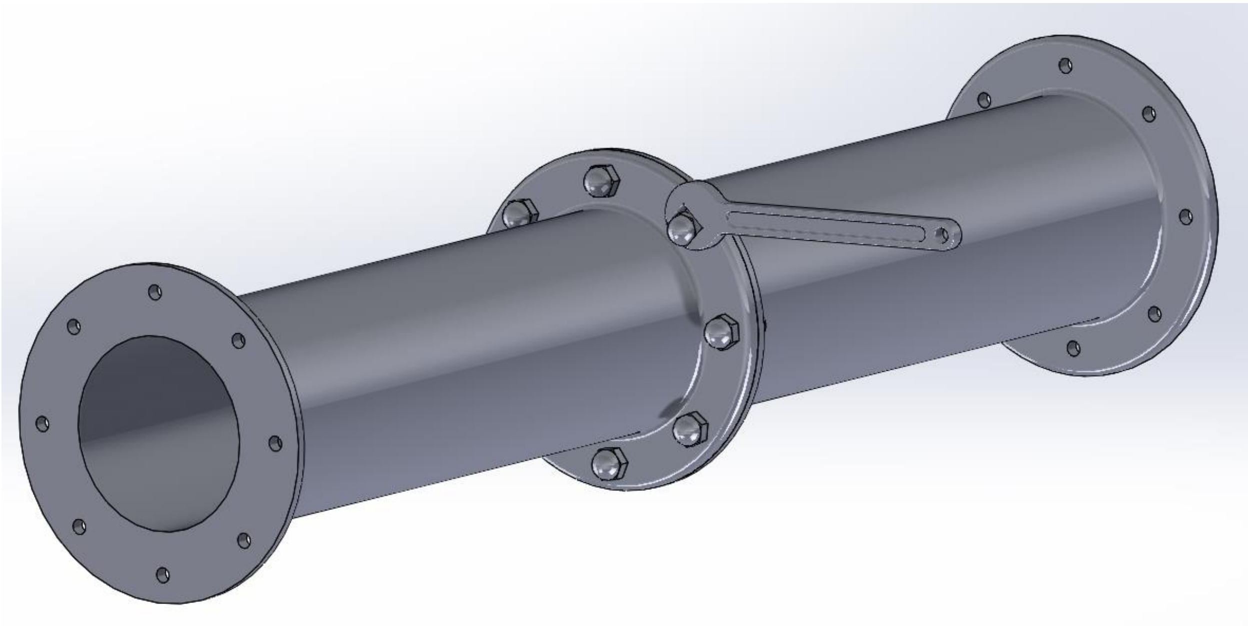
6. The final step is to apply some fillets to this pipe section. Add 5mm fillets to the inside and outside diameter of the base. Note again that if we applied this fillet before our mirror, we would only have to apply the fillets to one half of the section. Once we mirror the half section, the fillets would carry over to the mirrored half. It is important to keep these considerations in mind when using features such as Fillet, Mirror and Shell.



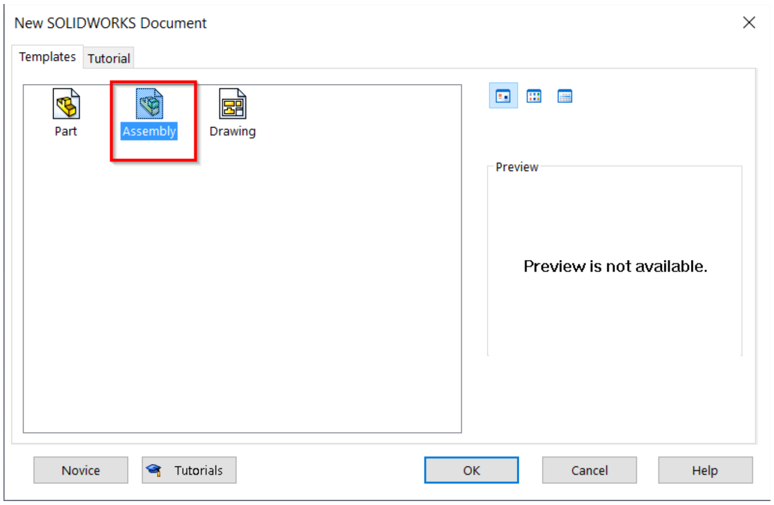
Exercise 4: Pipeline Assembly

Features Learned: Assemblies, Mating, Drafting Parts within Assemblies

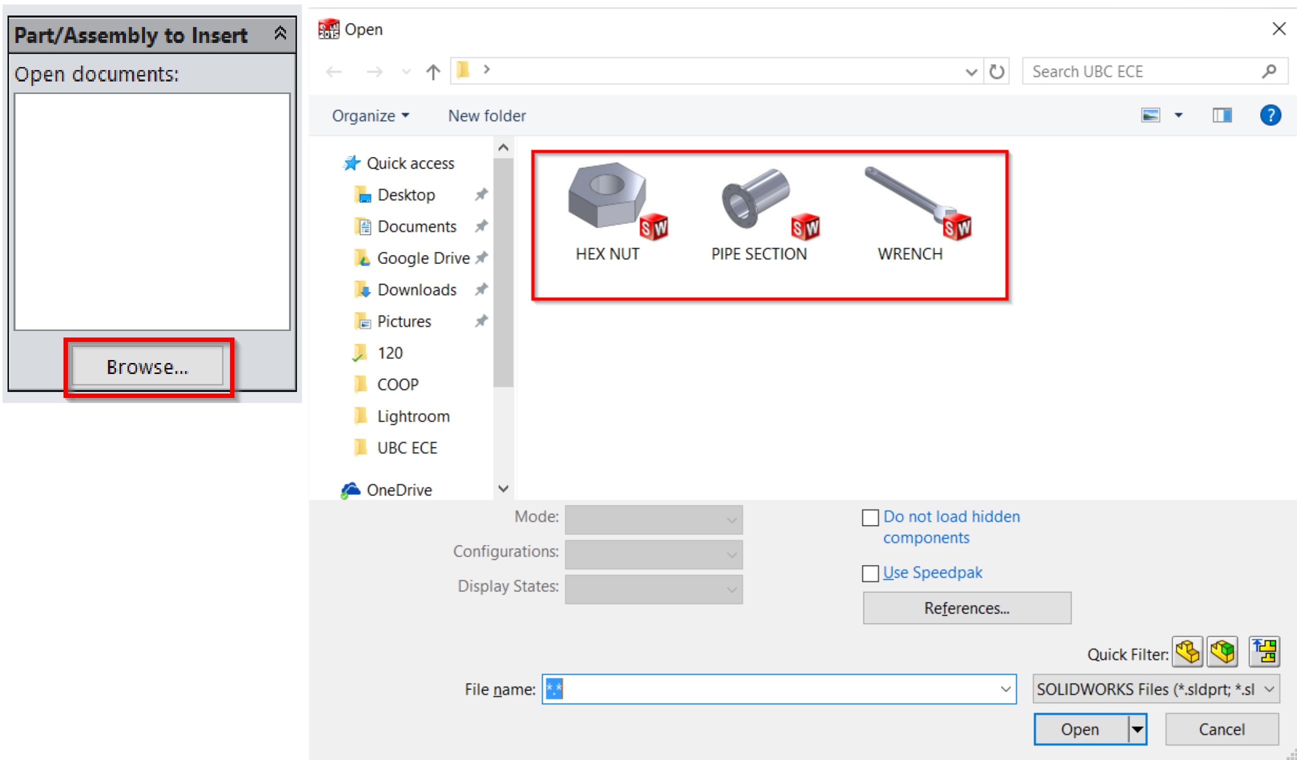
In this exercise we will be utilizing all the parts used in the previous exercises to assemble together a small length of pipe. The hex nut, wrench, pipe section will all be used. We will also design a very simple pin, which will be created directly within the assembly. This exercise will give a basic overview of assemblies and mates in SolidWorks.



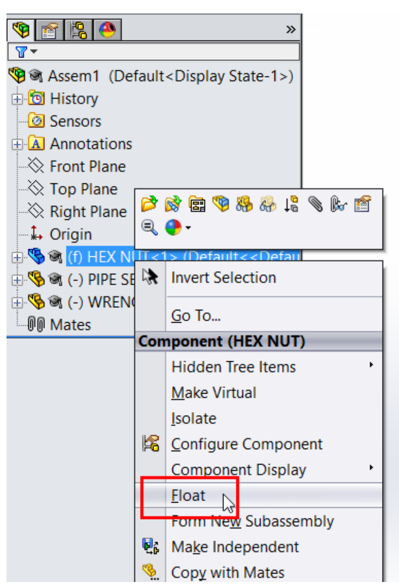
1. First, start a new assembly in SolidWorks. Up until this point we have been starting by making a new part. This time, start a new assembly. Be aware that the units in the assembly may be in IPS (inches, pounds, seconds). Make sure to change the units to MMGS in the bottom right hand corner before you begin.



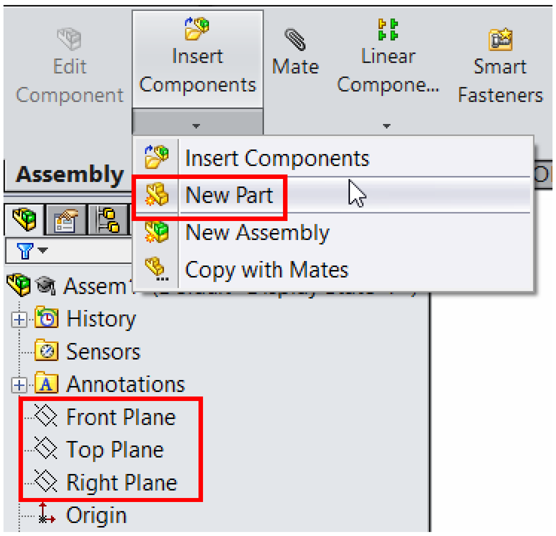
2. On the right hand side in the Part/Assembly to Insert panel, click the browse button and select the 3 components we have designed so far, the hex nut, the wrench and the pipe section.

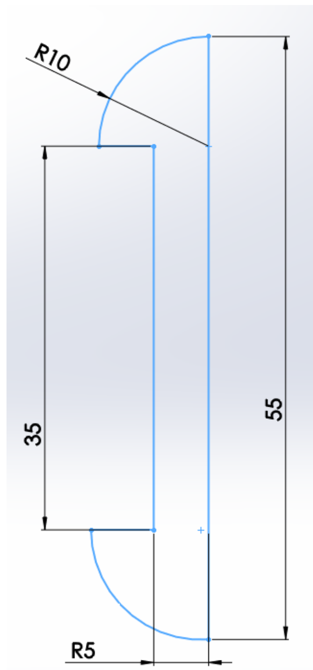


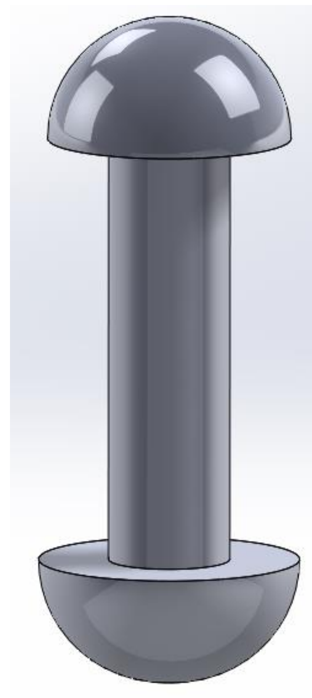
3. Once the parts are opened, drop them anywhere on the screen by left clicking. When the parts have been placed in the assembly, check to see that all parts are free to move by clicking and dragging them. If one of the parts does not move (it is fixed) then find the part in the left hand panel, right click, and click float. The (f) in brackets next to the part name signifies that it is fixed and cannot move.



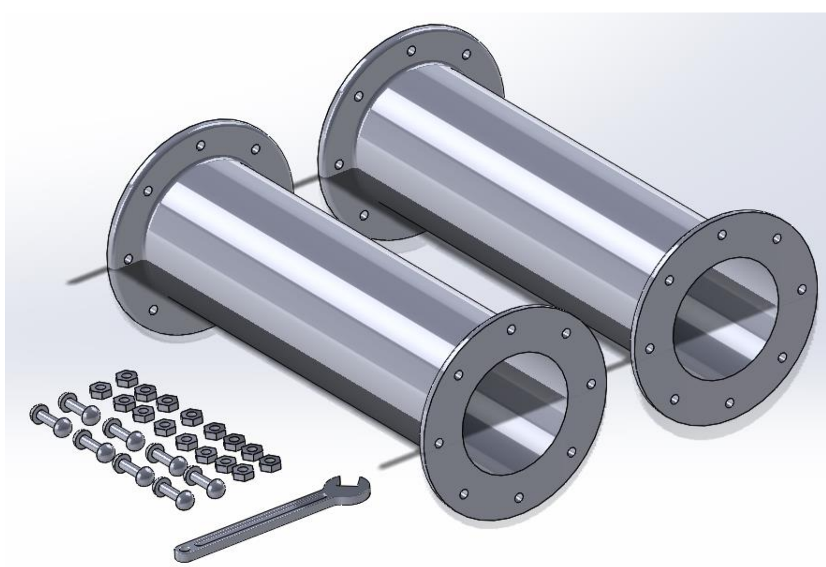
4. After all parts have been placed in the assembly it is time to design the final part, the pin. We will be drafting this part directly within the assembly. First click the drop down menu under Insert Components and select New Part. Select any of the principle planes to sketch on. At this point we can sketch and draft a part in the same manner we have done previously. Sketch the following outline and use the Revolved Boss/Base tool to rotate the sketch 360° to obtain a solid part. Be sure to unselect Edit Component to finalize the part.



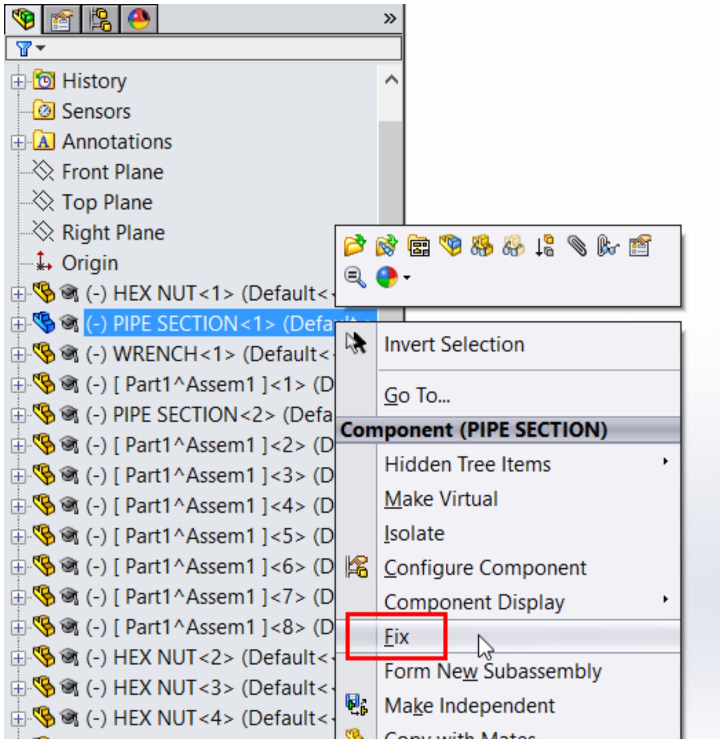




5. Once the pin is designed within the assembly we have all parts necessary to construct our pipeline! One of each part should already be in the assembly from step 2. To copy parts, simply click and drag a box around the part you wish to copy until the entire part turns blue, and CTRL + C and CTRL + V as many copies as needed. For this assembly we will need 2 of the pipe section, 16 hex nuts, 1 wrench and 8 pins.

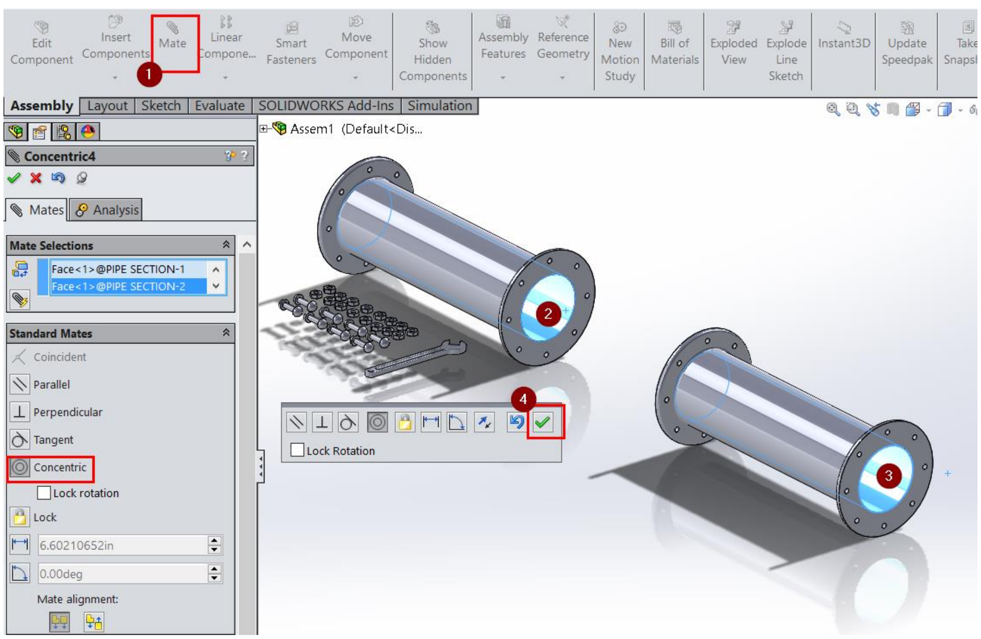


6. Once we have the correct number of parts we are ready to assemble our pipeline. It is helpful to fix one of the pipe sections so it doesn't move, and attach all the other parts to the one fixed pipe section. Right click one of the pipe sections in the panel on the left and click fix.



7. Once one of the pipe sections is fixed we are ready to mate the other components to it. Mating is a tool used in assemblies to attach components together.

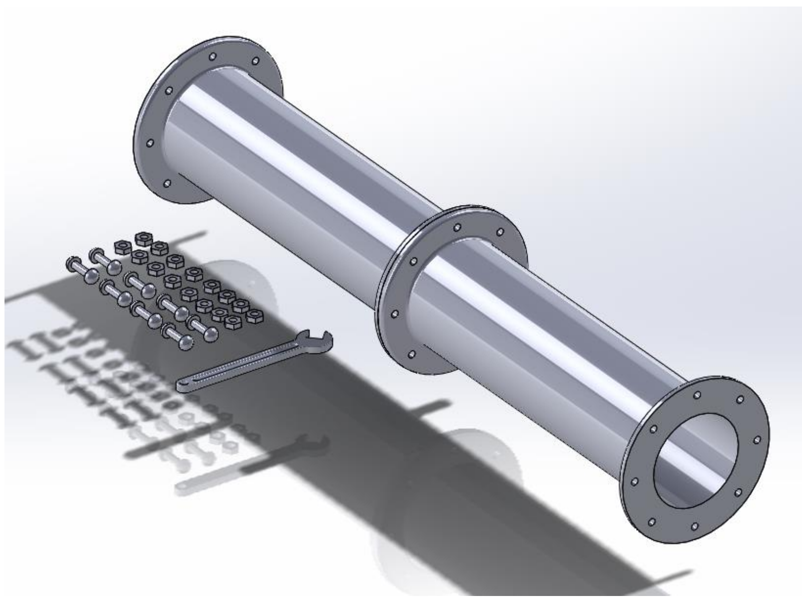
We will start with the Concentric Mate. Start by selecting the Mate tool in the Assembly tab at the top. Then select both inside surfaces of the two pipe sections.



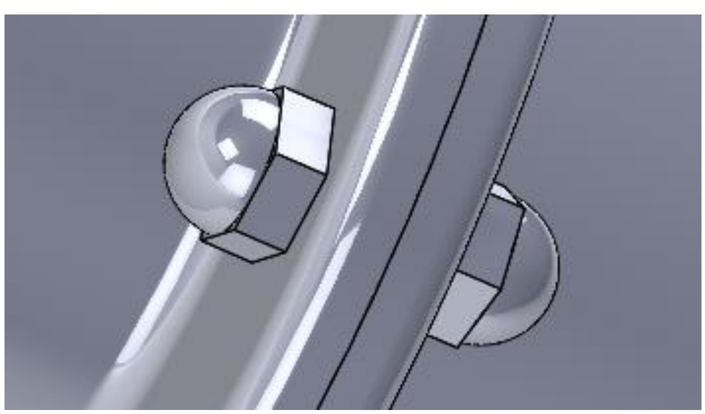
Note how SolidWorks recognizes the surfaces to be mated and automatically selects the Concentric Mate. There are many different mates as seen in the Standard Mates panel. In addition to the standard ones, there are Advanced Mates and Mechanical Mates, which are more complex and have more parameters.

Now that the two pipe sections have been concentrically mated, notice how if you try to move the unfixed section, it only moves along one axis and rotates about that axis. To completely fix the two sections together, we need to add two more mates, one to fix the components axially, and another to fix the components rotationally.

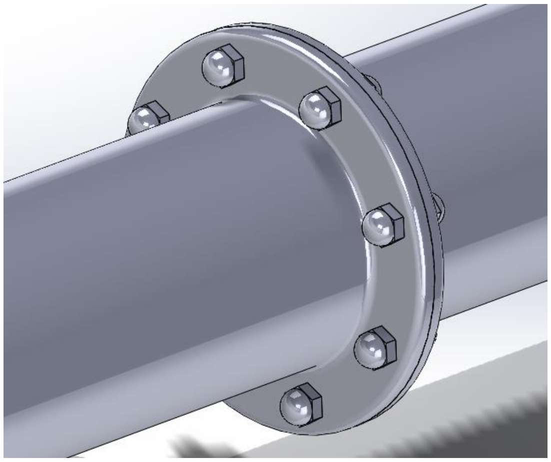
8. Mate the two base surfaces of the pipe sections. SolidWorks should automatically set a Coincident mate. Next we have to fix the sections together rotationally. To do this, mate two of the bolt holes on each pipe section with a concentric mate. Now the two pipe sections should be completely fixed with no degrees of motion.



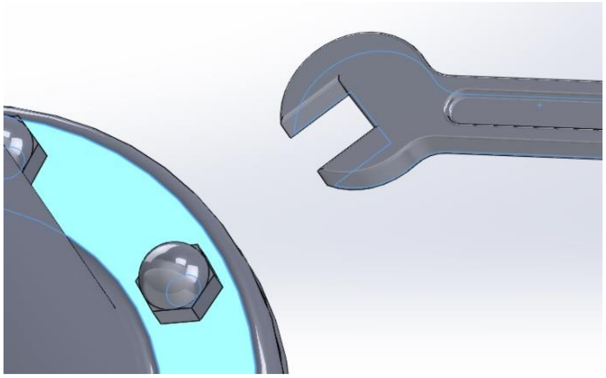
9. Mate the nut/pin assembly as shown. For each bore, mate two nuts and one pin using a series of standard mates (coincident, concentric). These mates are fairly uncomplicated and should be done with relative ease.

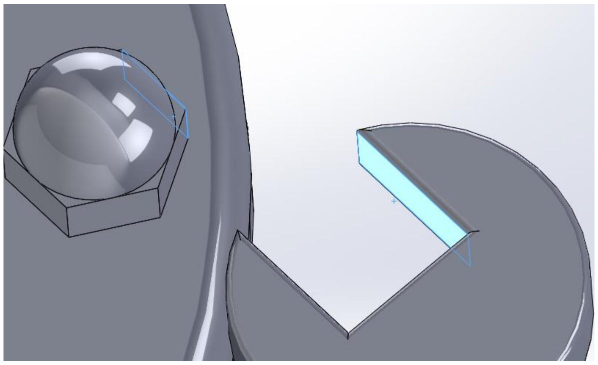


10. Repeat this process for all 8 bores, using all 16 nuts and 8 pins in the process. The assembly should appear as follows. Take note: the nuts and pins are still able to rotate freely. It is possible but not required to apply a lock mate (under standard mates) to the nuts and any surface of the pipe section to ensure that they are not free to rotate.

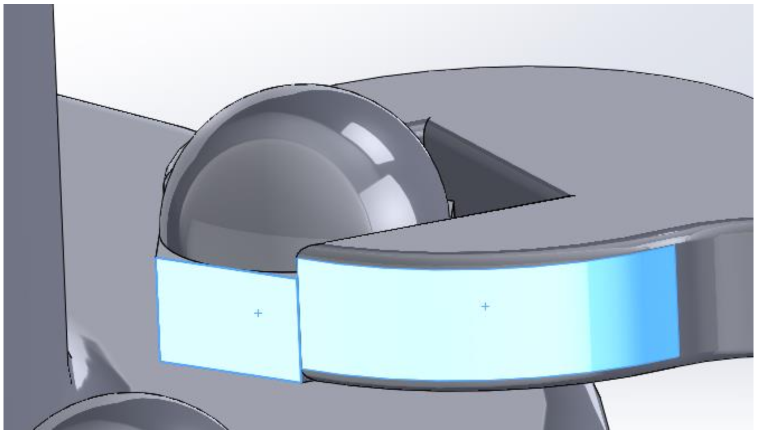


11. The final step is to mate the wrench to one of the nuts. First, mate the bottom surface of the wrench to the surface of the pipe section base. Next, mate the top inside surface of the wrench to one of the sides on the nut.





Finally, mate the bottom curve of the wrench head to the corresponding side of the nut. The wrench should now be fully mated to the nut. The assembly at this point is finished.



Exercise 5: 3D Printing with the Tinkerine Ditto Pro

In this exercise we will explore 3D printing SolidWorks parts using the Tinkerine Ditto Pro. These 3D printers are available for ECE students enrolled in project courses, ECE research work, and other related UBC research.

Setup and Calibration

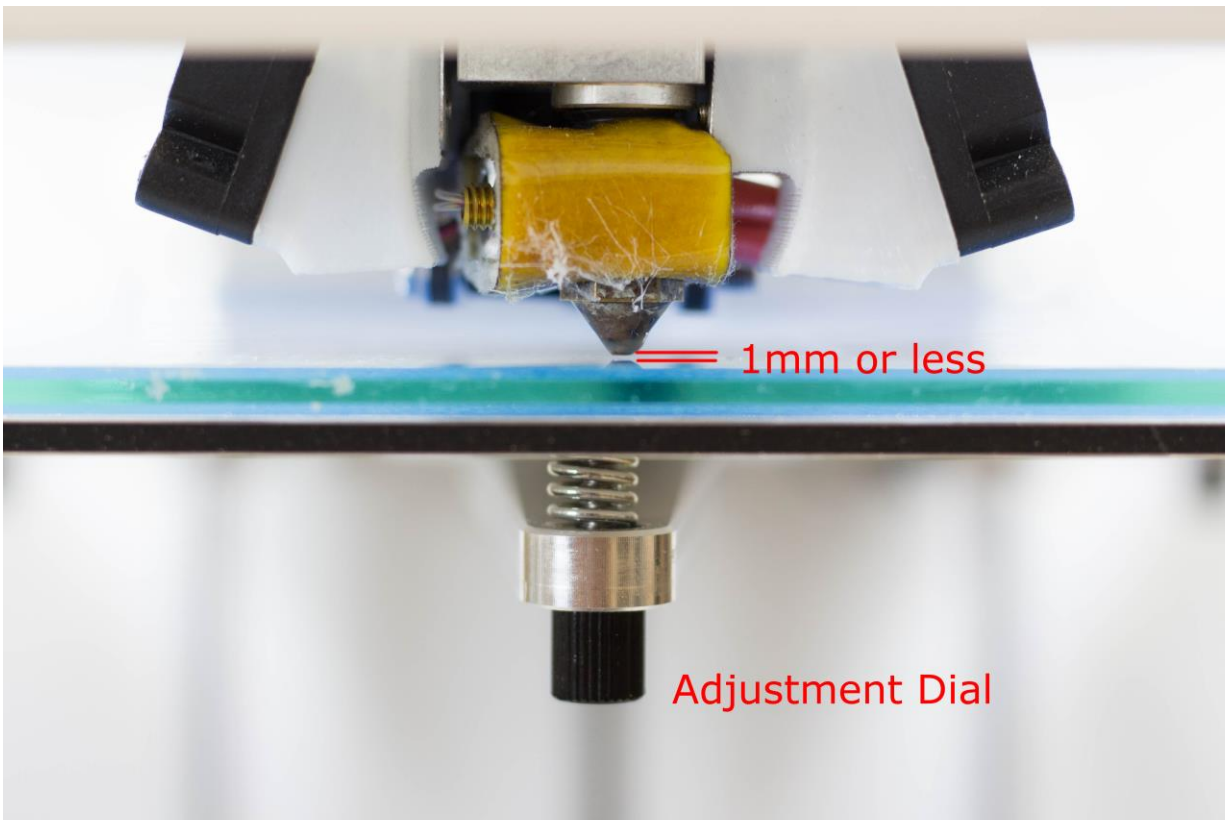
The first thing to do before 3D printing an object is to calibrate and setup the Tinkerine Ditto Pro 3D printer. Setup and calibration consists of 3 steps: leveling the printing bed, loading new filament, and preparing the build platform

1. Levelling the Printing Bed

This calibration wizard is used to make sure the printing bed is level. A level bed is important since any significant angles to the bed may cause malfunctions and misalignment during printing. To activate the bed level wizard, ensure that the Tinkerine Ditto Pro 3D printer is plugged in, and the power turned on.

Navigate to the Wizard menu using the scroll wheel. Once in the wizard menus, select Bed Level.

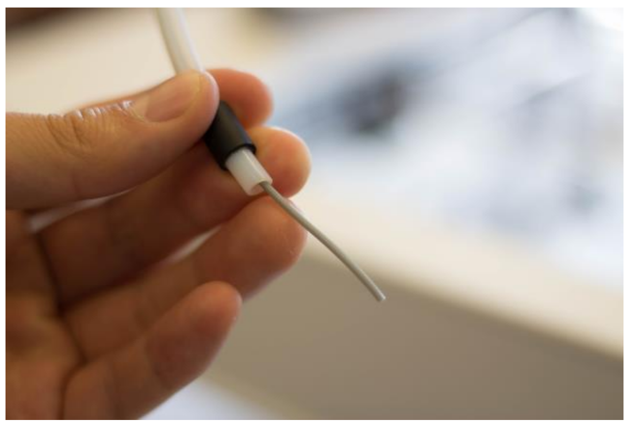
Follow the instructions on the display, ensuring that a small space is left between the printer head and the build platform at every point (about 1mm or less). To change the distance between the printing bed and the printing head, adjust the small black dials on the bottom of the bed.



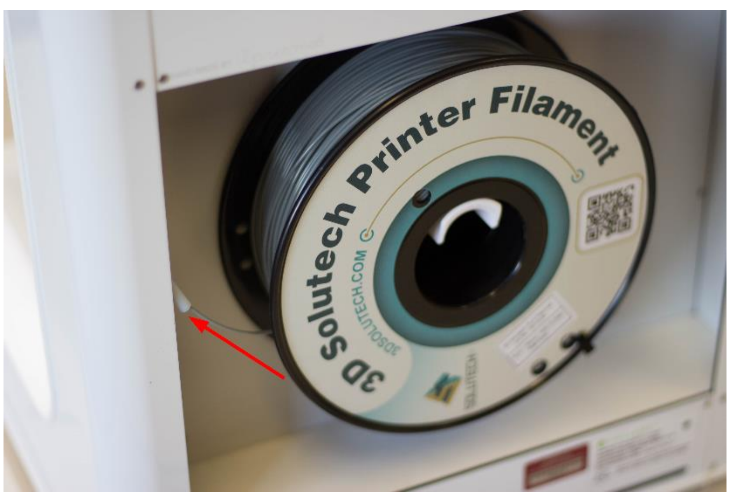
2. Loading New Filament

This calibration wizard is used when loading new filament into the printing head. In the wizard menu, select New Filament. Follow the instructions on screen. You may need to wait for the printing head to heat up.

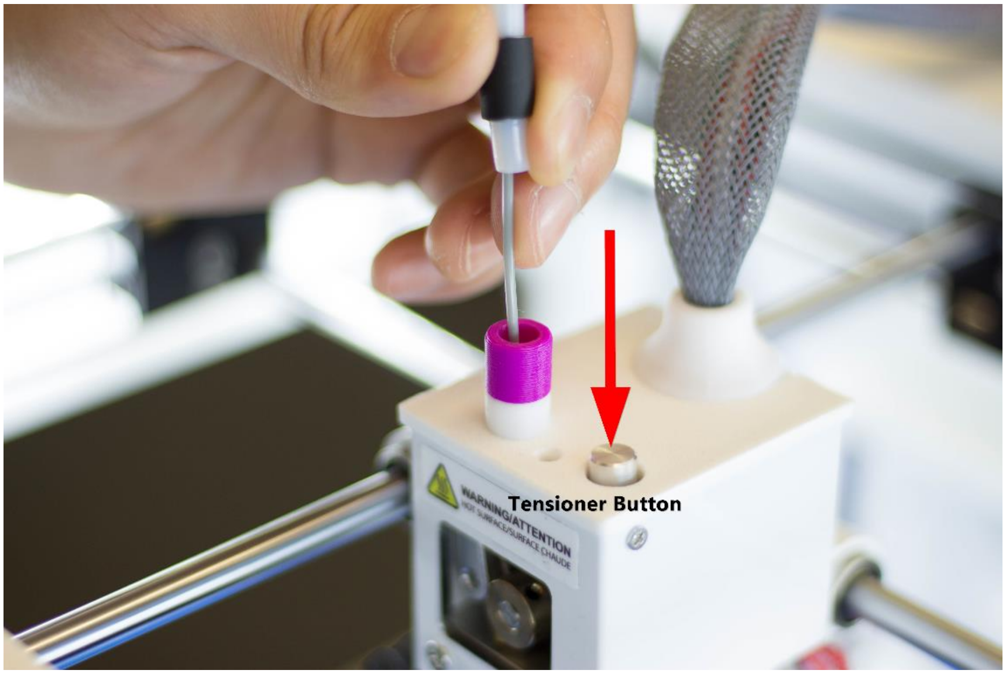
Remove any old filament and feed the new filament up through the guide tube and out the other side.



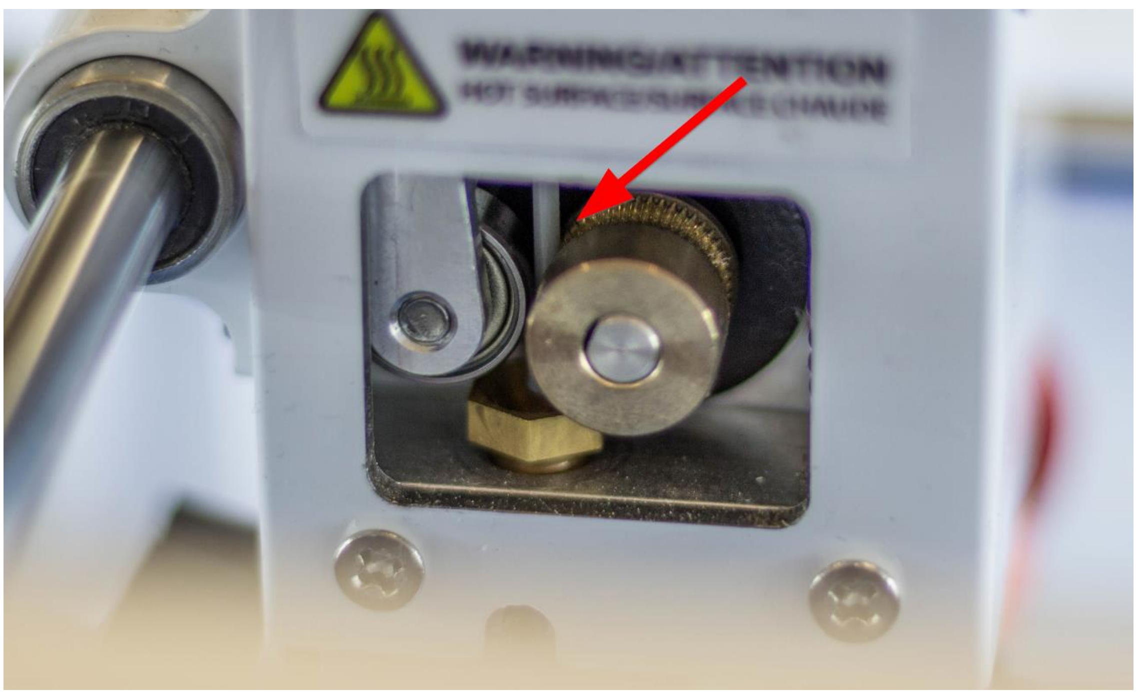
Ensure that the spool is hung neatly on the back of the printer, with the filament being spooled out clockwise and into the guide tube as shown.



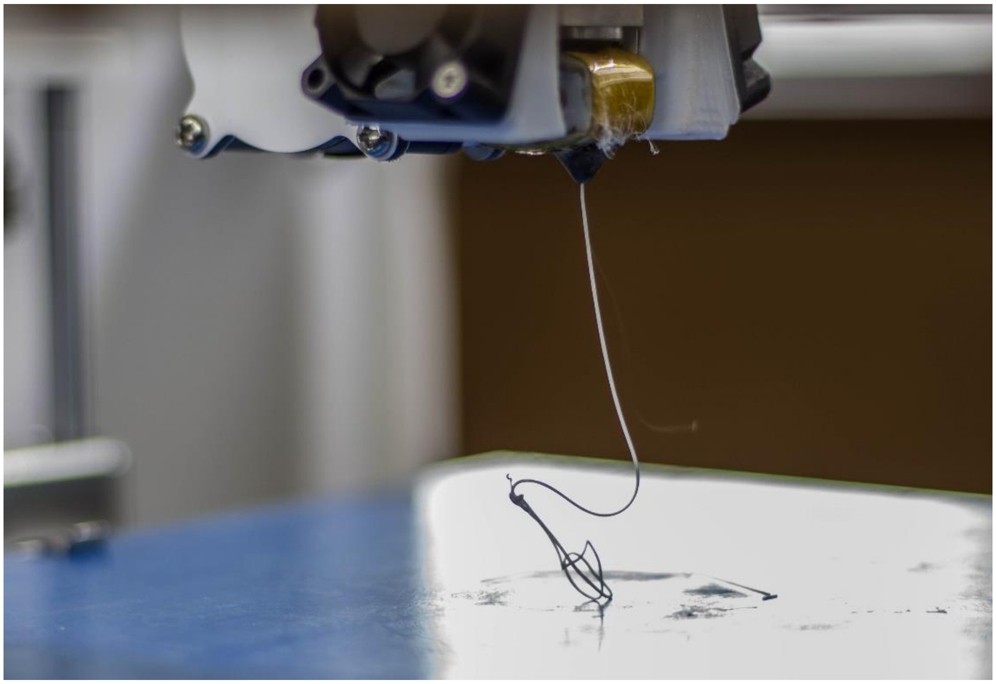
Press the tensioner button down to thread the new filament into the hot end of the printer head.



Note the position of the filament, make sure it is fed in between both gears.



Push the filament through the head until it comes out of the hot end. Trim off the excess filament. If the printer was previously loaded with a different colored filament, push the new filament through until you see the new color come out of the hot end.

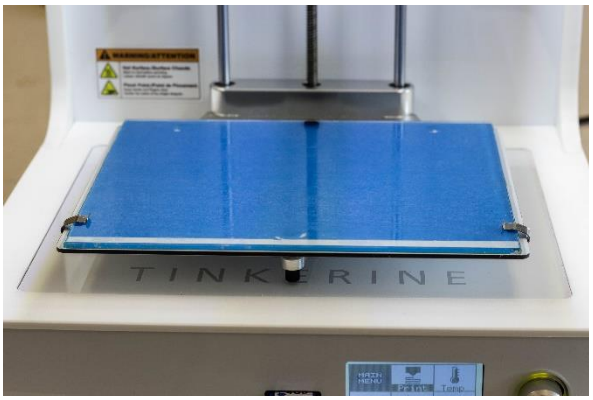


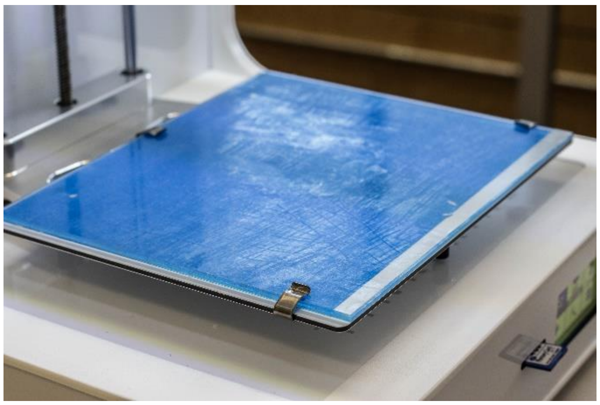
3. Preparing the Build Platform

First, while the printer is off, scrape off any dried glue or filament from the built platform using the tools provided.

Be sure to use the glass panes provided as the build platform, never print directly on the printing tray of the Tinkerine Ditto Pro. Once the built platform is clean, apply glue over the surface of the build platform. The glue will ensure that the filament sticks to the build platform.

Finally, ensure that the glass is firmly attached to the printing tray with the provided clips.





Preparing the .STL File

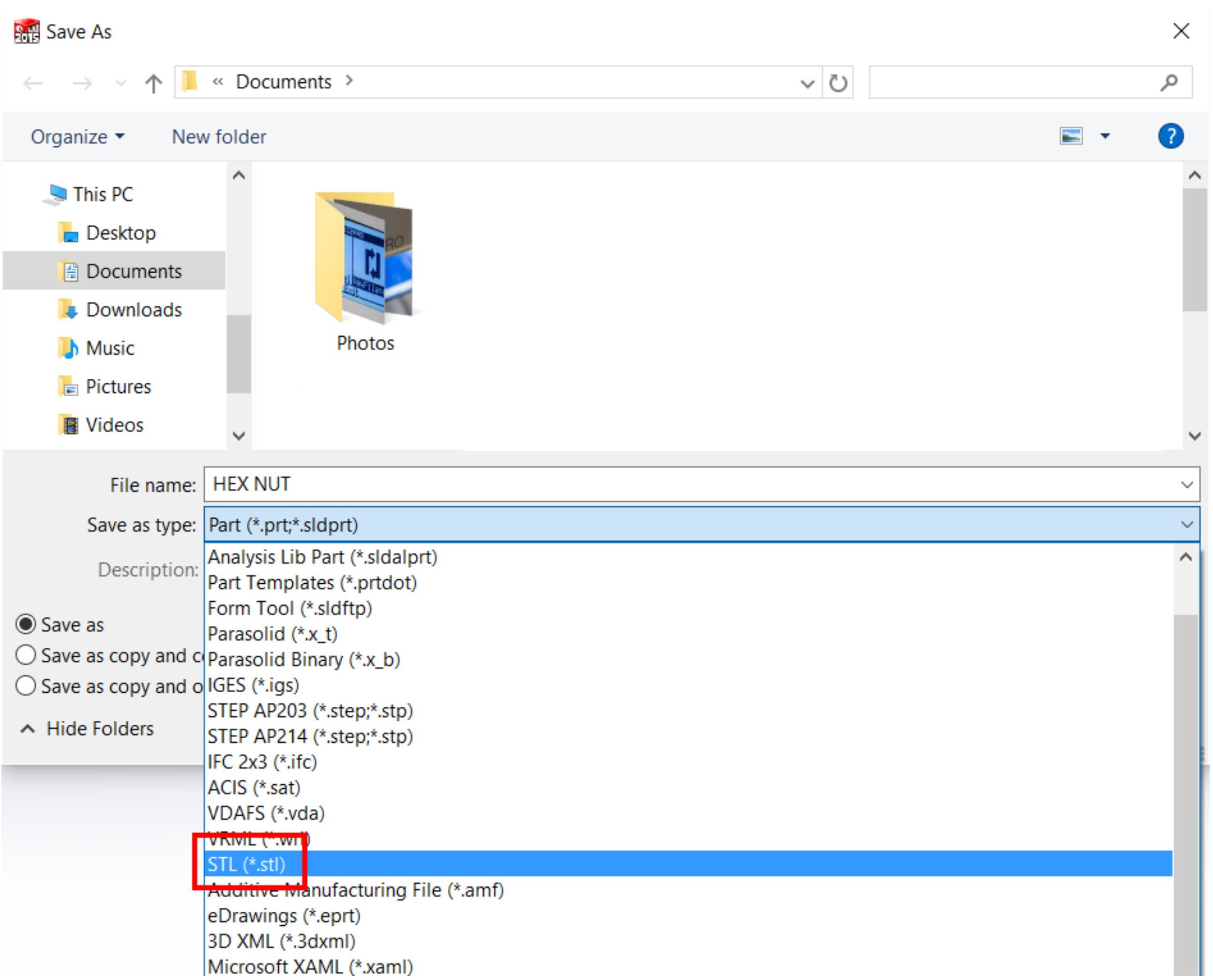
After the 3D printer is setup and calibrated, it is time to prepare our file for printing. After the part has been drafted in SolidWorks, it must be saved as a .stl file, then converted and sliced into a .g file before it is ready to be read and printed by the Tinkerine Ditto Pro.

Before beginning the process, first install the 3D printer slicing software. Tinkerine provides their own slicer program called Tinkerine Suite for free, it can be installed at the following link:

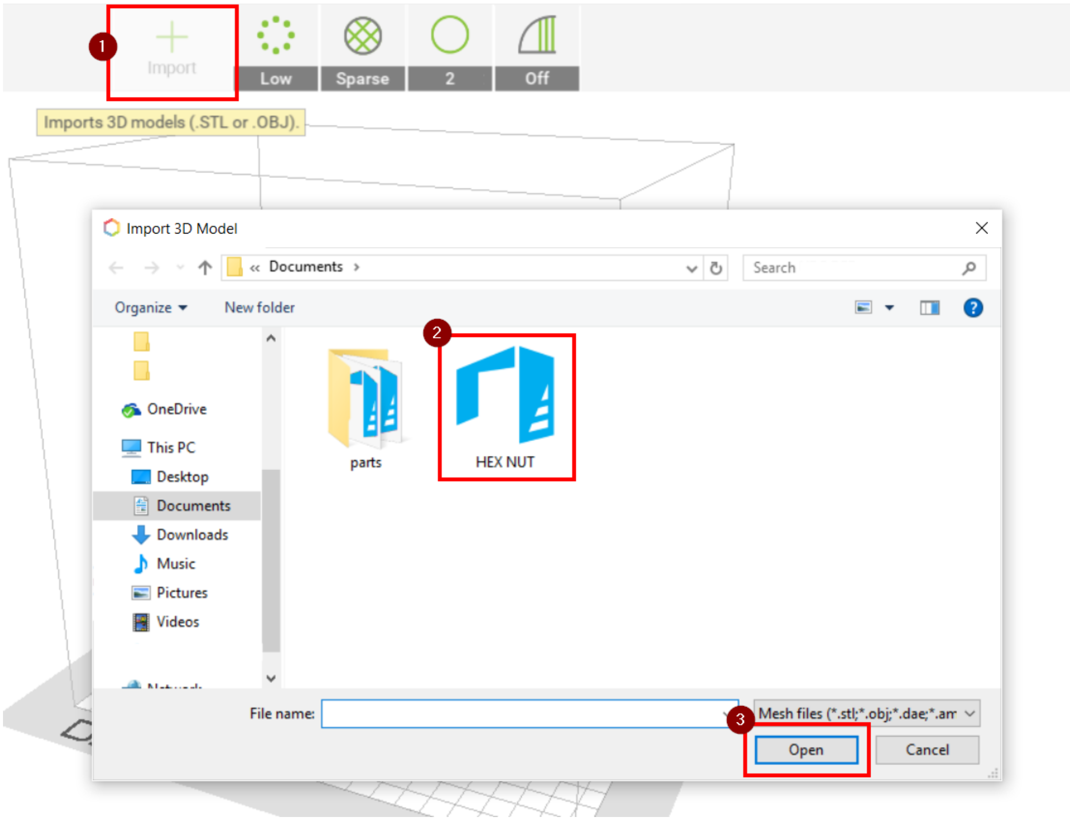
http://tinkerine.com/tinkerine-suite/

Once the slicer program has been installed, proceed to the next steps:

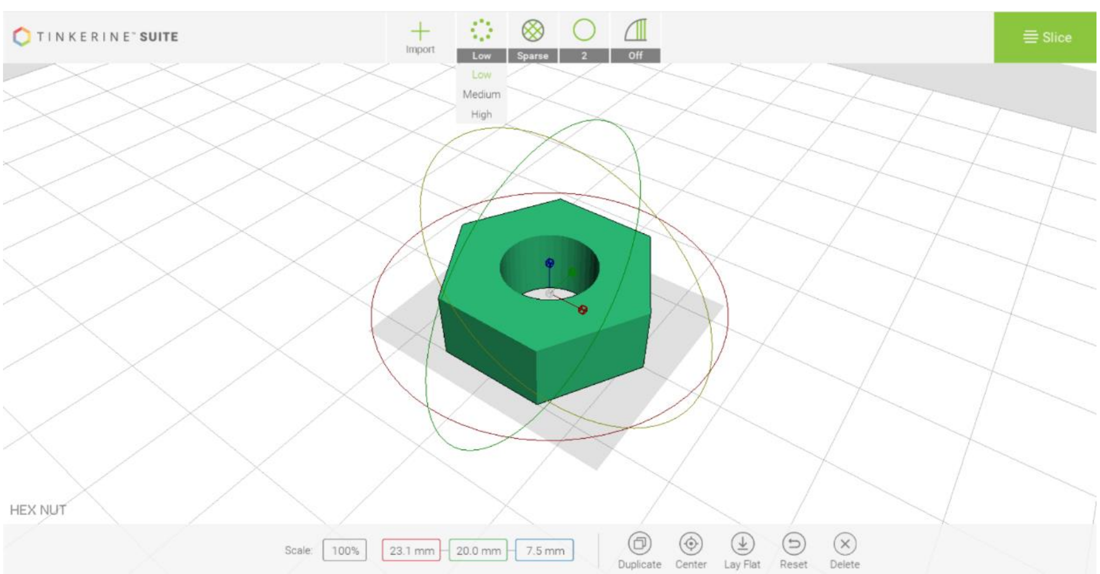
1. First, save the SolidWorks part as a .stl file. Select Save As, and choose .stl as the file type. STL is a file type that is universally used by most 3D printers and slicing software. It is the first step in preparing the SolidWorks part to be read by the 3D printer.



2. Once the part has been saved as an .stl file, we are ready to create a .g file using a slicer program. Open up Tinkerine Suite. To prepare our .stl file for printing, first import the .stl file into the program.



3. Once the file has been imported into Tinkerine Suite, the part should appear on the virtual printing bed. At this point there are a number of different options we can tweak, including Resolution, Infill, Wall, Support, Scale, etc.

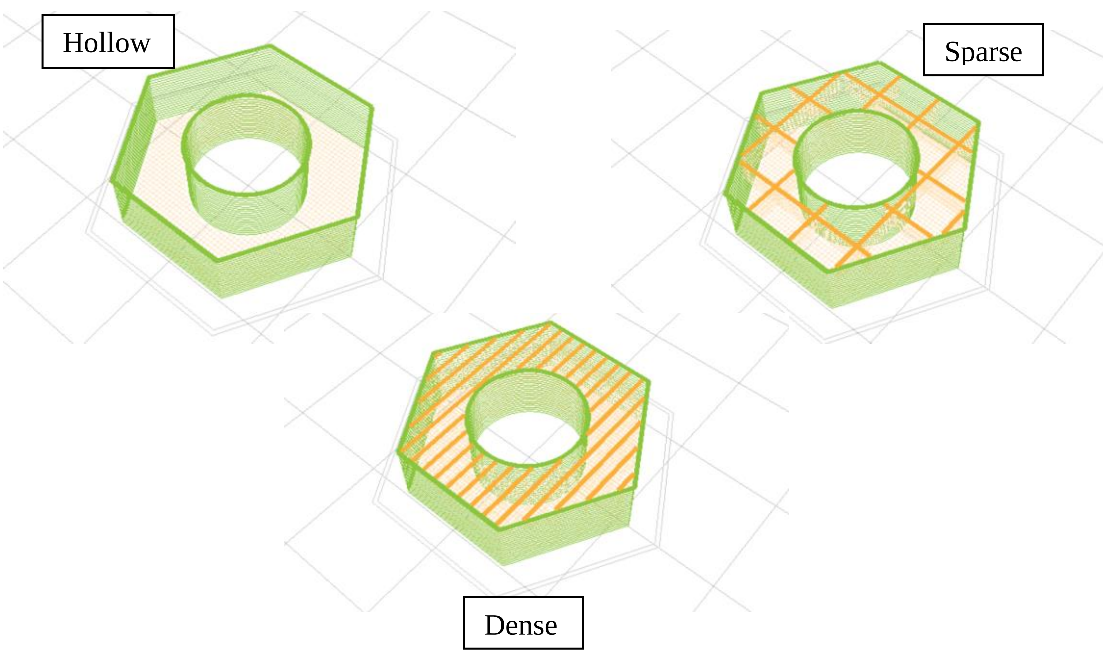


4. The resolution parameter controls the accuracy and precision of the printed material. The higher the resolution, the more precise the printing will be.

Resolution Level **(All other parameters unchanged)**	**Print time and filament used**
Low	Print Time 00h:03m Filament Usage 1g
Medium	Print Time 00h:05m Filament Usage 1g
High	Print Time 00h:09m Filament Usage 1g

Notice that the higher resolution comes at the sacrifice of printing time. A low resolution part will print much faster than a high resolution part. Generally, for simple parts without complex details (such as our hex nut), low resolution is adequate.

5. Infill describes the how the solid components are filled. Hollow means that there is no material inside the solid components of the part. Sparse will print a grid of support material inside the solid parts, and Dense will print a more compact matrix of support material. Similarly to resolution, a denser infill will increase printing time, but it will also add to the mass of filament used. Generally it is a good idea to have at least some infill, as hollow parts will be prone to failure during and after printing.

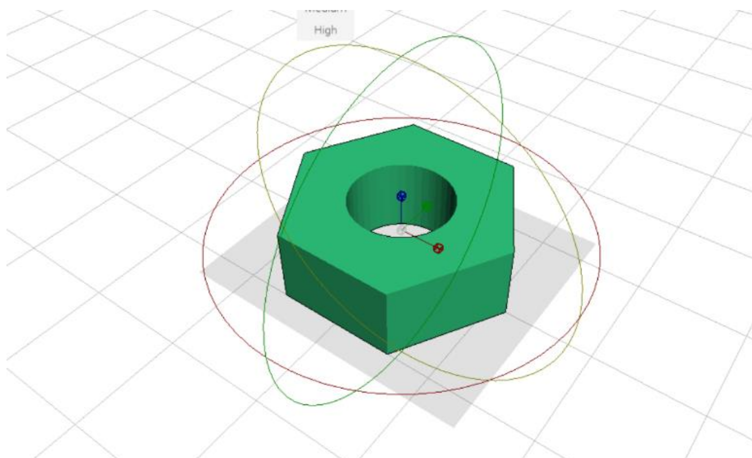


6. Wall is a parameter that controls how thick the printed walls will be of hollow/infilled components. Thicker walls translate to higher structural integrity but again comes at the cost of printing time and mass of filament used.

7. The Support parameter controls whether or not support material will be printed. Support material is an important consideration when dealing with overhanging components of a part that is to be 3D printed. When parts are overhanging, there is the potential for the part to not print properly. This phenomenon is especially prevalent when the angle of overhang exceeds 60° with the horizontal. In these scenarios it is good practice to print with support material but again, the print time and filament used will be increased as a result.

8. Once the desired parameters have been set it is imperative to consider the orientation of the part. Since the Tinkerine Ditto Pro 3D printer prints in layers of melted filament, the orientation of the printed part will have a tangible impact on printing time as well as the mechanical properties of the part. It is good practice to always orient the part such that circular holes/bores are oriented up/down instead of through sideways. Also, a flat/wide orientation is preferred to a tall/skinny orientation. The figure below shows the most ideal orientation for the hex nut we designed earlier.

9. Prepare file as a .g file by clicking the “Slice” button. Save this .g file to an SD card. We are now ready to print our part



Printing the .g File

After the 3D printer has been calibrated and the .g file prepared, it is finally time to print our part.

Simply insert the SD card into the Tinkerine Ditto Pro. Power on the device and navigate to the print menu. Find the file to be printed and push the button to begin the printing process. After a brief heat up period, the printer should start printing the part. Be sure to monitor the printing carefully and look out for any malfunctions by the printer itself or the filament spool. Once the part has finished printed, remove the part using the scraper.