SOLIDWORKS Lab

GNG1103 – Engineering Design

Objective

The objective of this SOLIDWORKS lab is to learn the basics of 3D modeling. This can be useful for a number of reasons related to design, including prototyping, simulation and analysis. In this lab, you will learn the basic functions of SOLIDWORKS which will allow you to create 3D parts and assemblies. You will see how to save parts in a file format which is compatible with 3D printing, another useful prototyping tool. Please note that using a mouse with SOLIDWORKS is essential

SOLIDWORKS

SOLIDWORKS is a solid modeling computer-aided design (CAD) and computer-aided engineering (CAE) computer program. Computer-aided design (CAD) is the use of computer systems to aid in the creation, modification, analysis, or optimization of a design. CAD software is used to increase the productivity of the designer, improve the quality of design, improve communications through documentation, and to create a database for manufacturing. CAD output is often in the form of electronic files for print, machining, or other manufacturing operations. The output of CAD must convey information, such as materials, processes, dimensions, and tolerances, according to application-specific conventions. Computer-aided engineering (CAE) is the broad usage of computer software to aid in engineering analysis tasks. It includes finite element analysis (FEA), computational fluid dynamics (CFD), multibody dynamics (MBD), and optimization, all of which are extremely useful in creating better designs.

Pre-Lab Questions

What is a CAD software?

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What can a CAD software do?

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What is the difference between a part and an assembly file?

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What file format do you need to be able to 3D print?

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What is a 3D printer?

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Accessing SOLIDWORKS

SOLIDWORKS is freely available to all uOttawa engineering students and can be accessed through the uOttawa Faculty of Engineering Remote Applications System. To log into Remote Apps, go to:

http://remoteapps.genie.uottawa.ca

(Mac users can follow these instructions to use Remote Apps: http://www.site.uottawa.ca/rdg/Mac.html)

*Note: If you are in a lab room and can use a faculty computer do **NOT** access SOLIDWORKS through remote apps, instead run the application on the computer itself. If you are using your own personal computer however then use remote apps.

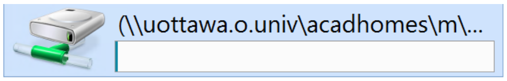
Once you have logged in, follow the instructions below to use SOLIDWORKS:

1. On the main page, click on “SOLIDWORKS 2015 x64 Edition”
2. Choose OK when the “Open with: Remote Desktop Connection” window pops up
3. Choose Connect when Windows asks “Do you trust the publisher of this RemoteApp Program?”
4. Enter credentials again using your full uottawa email address
5. SOLIDWORKS may take a few moments to load. Be patient!
6. Accept “SOLIDWORKS License Agreement”
7. Accept “Simulation License Agreement”
8. Choose OK for “I am a new user. Show Quick Tips to help me get started.”

You now have an empty SOLIDWORKS window open in front of you. If your internet connection is interrupted (or you mistakenly close the remote app), you have ***ONE HOUR*** to log back on and resume your session right where you left it. After an hour of inactivity, the server will reset automatically, and you will have to start the procedure all over (and will lose your work). It is good practice to save your work frequently! The easiest way to log back on to the ***same*** SOLIDWORKS session is to click on the file which was downloaded with your browser the first time you clicked on SOLIDWORKS.

If you have any issues whatsoever login on and using Remote Apps, please see the Remote Apps help document “Remote Apps Guide v2.3.pdf” located in the lab folder on Virtual Campus.

Opening and Saving Files- Z:



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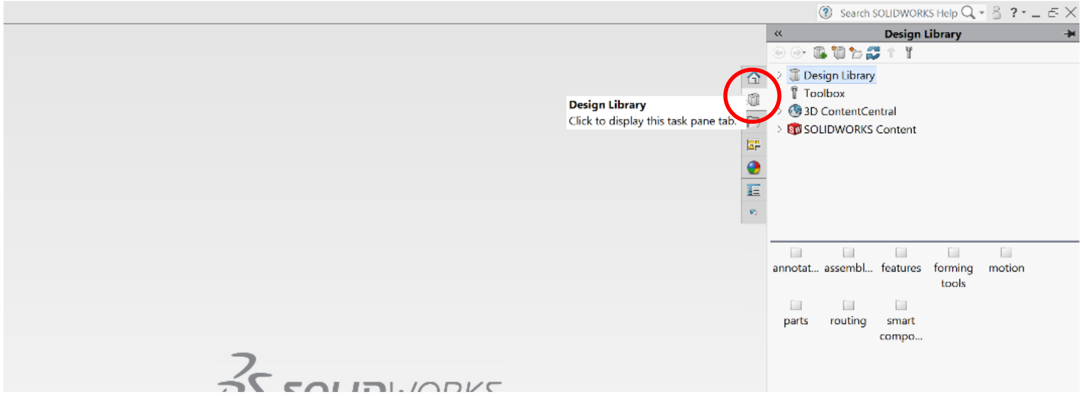
Because SOLIDWORKS is running as a remote application, ***IT IS STRONGLY RECOMMENDED*** that you open, use and save files exclusively from your Z: drive on the Faculty of Engineering servers while using SOLIDWORKS. Files can be copied to and from your computer to the Z: drive using the Remote Apps application by copying and pasting files in the SOLIDWORKS “Open” window from your computer (C: drive) to your Z: drive. You will see a significant decrease in performance if you attempt to use files located on your computer. Any files saved to the server itself using one of the regular documents, downloads, desktop, etc. folders will be ***deleted*** when you log off the system.

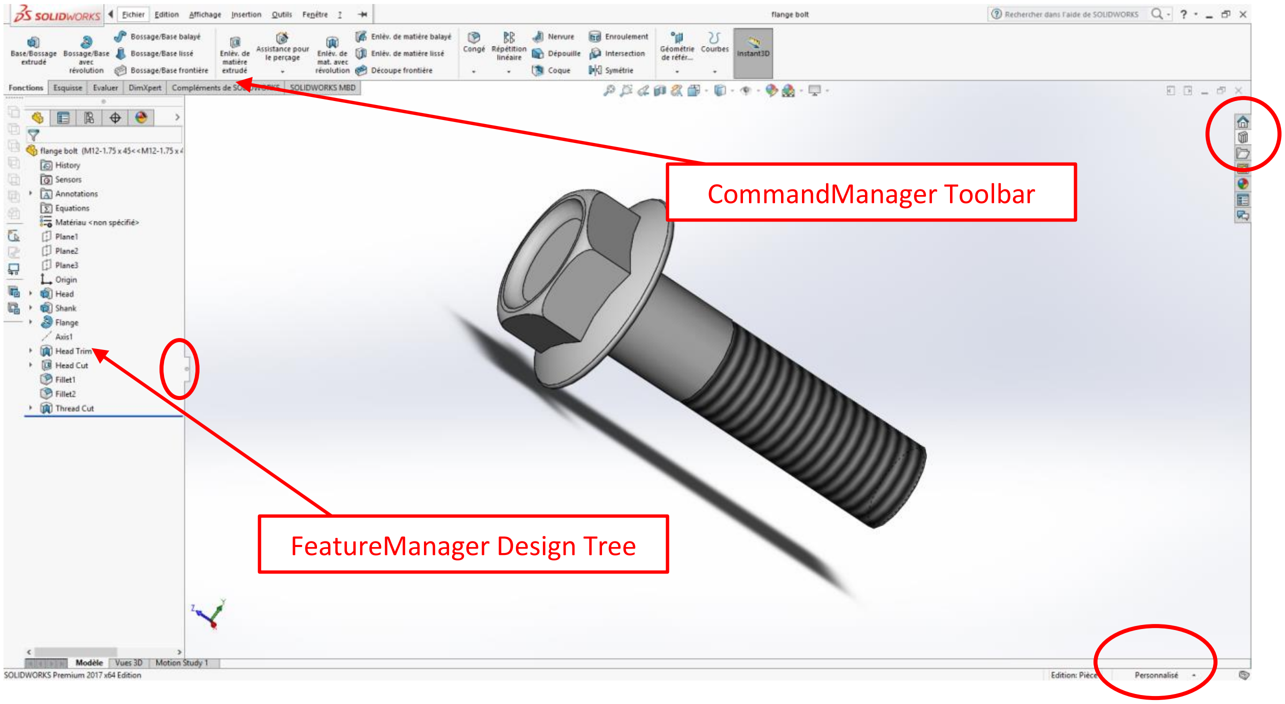
SOLIDWORKS WINDOW

Click on the “Design Library” icon on the right-hand side of the SOLIDWORKS window as circled in red below:



Click on “Design Library” in the list that appears. Double-click on the “parts” folder that appears. Double-click on the “hardware” folder that appears. Double-click on the “flange bolt” that appears.

You may need to expand the “FeatureManager Design Tree” by clicking on the left-hand side of the window as circled below, but your SOLIDWORKS window should now look like this:



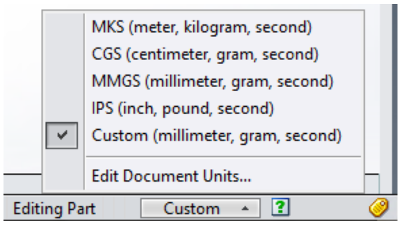
CommandManager Toolbar

FeatureManager Design Tree

The left-hand menu is known as the “FeatureManager Design Tree”. This list of items will be constantly updated as you work on your model. As you add features to your model, extra lines will be added to the list. To edit models, you can click and right-click listed features to access various options for these features.

Try right clicking on the “Head” feature in the “FeatureManager Design Tree” and then click on the edit sketch icon (second one from the top-left in the menu). What happens?

TIP: You can change the default units used in SOLIDWORKS (in this case metric) by clicking on the menu in the bottom right-hand corner of the window as circled in red in the image above. The resulting menu should look like the one in the image to the right. Try selecting IPS so that "inch" is used as the default unit.


MKS (meter, kilogram, second)
CGS (centimeter, gram, second)
MMGS (millimeter, gram, second)
IPS (inch, pound, second)
✓ Custom (millimeter, gram, second)
Edit Document Units...
Editing Part
Custom

Left click on the "Head" feature again. What happened to the value representing the width of the bolt head?

Click on the big red "X" in the top right-hand corner to close the sketch. Then click on "Discard Changes and Exit".

Mouse Buttons

Mouse buttons operate in the following ways:

Left – Selects menu items, entities in the graphics area, and objects in the FeatureManager design tree.

Right – Displays the context-sensitive shortcut menus.

Middle – Rotates, pans, and zooms the view of a part or an assembly, and pans in a drawing.

Try rotating the bolt around by clicking on the middle button (scroll wheel) and moving the mouse around. Try zooming in and out by rotating the middle scroll wheel. What happens if you hold down the right mouse button and move the mouse around?

Close the "flange bolt" part model by clicking on the lower of the two "X" in the top right corner, as circled in red on the image above. Then click "Don't Save".

Tips

1. Using a mouse with SOLIDWORKS is essential. Using a trackpad on a laptop is very difficult due to the lack of a scroll wheel for moving and rotating the view.
2. When entering dimensions, there is no need to convert dimensions which aren't the same as the default dimensions. If your drawing is in "mm" and the measurement you have is 2 inches, type

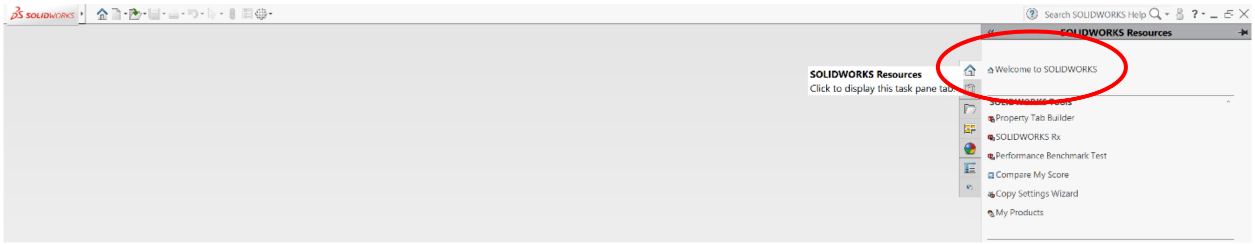
“2in” into the dimension “Modify” box and SOLIDWORKS will convert the value to “50.8mm”for you!

3. Proper sketches should always be “Fully Defined”. This means that all of the lines on the sketch are now black and you will not be able to move any of these lines by clicking and dragging them as their dimensions have been set. This can be done using the sketch dimension tool along with the on-screen constraints. What happens when you click and drag blue lines on a sketch?

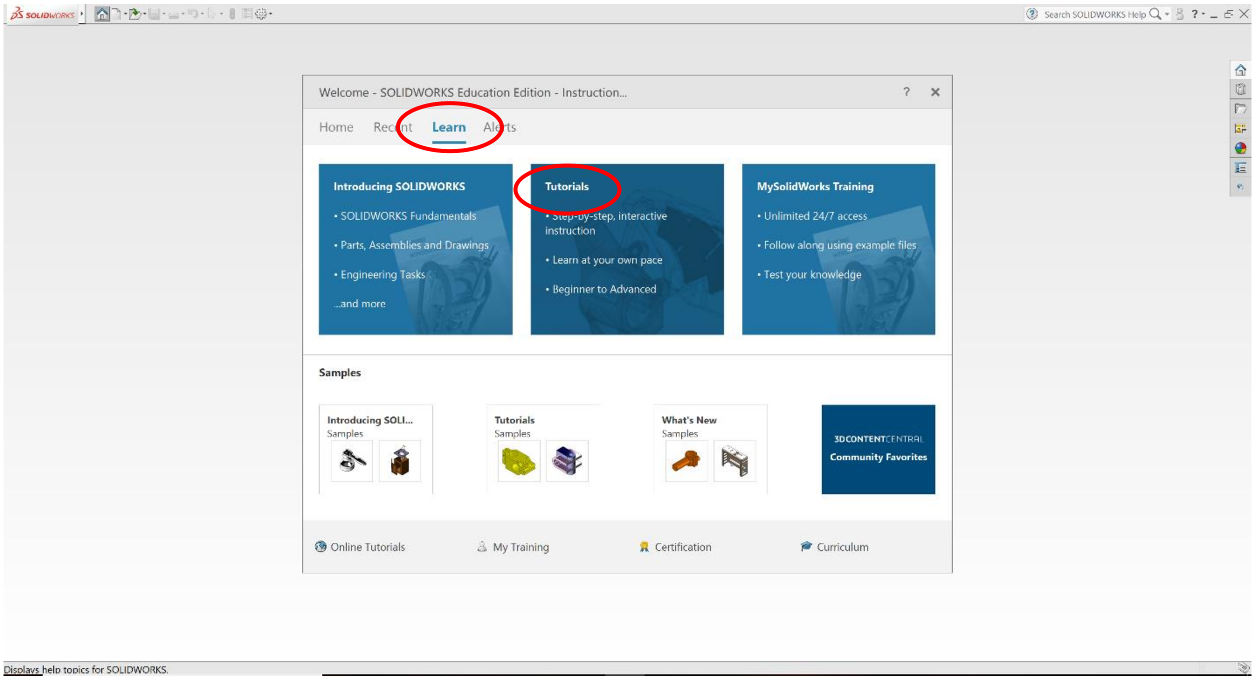
4. When creating a 3D model for manufacturing, square corners are unrealistic and ugly. Use the “Fillet” tool freely to round off sharp corners.
5. The best approach to creating solid models is to add or remove features as you would by making the part in a machine shop using actual machining tools. This helps in ensuring that you can actually make what you have designed!
6. If particular toolbars are missing, you can add them by right-clicking anywhere on the CommandManager toolbar. Particularly useful toolbars include: “Standard Views” and “View”.

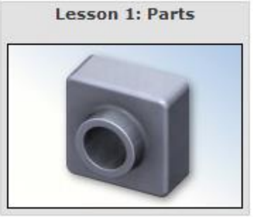
Tutorial 1: Creating a Part

You will now make a part using a SOLIDWORKS tutorial. Click on the “SOLIDWORKS Resources” icon and then on “Welcome to SOLIDWORKS” on the right-hand side of the SOLIDWORKS window as circled in red below:



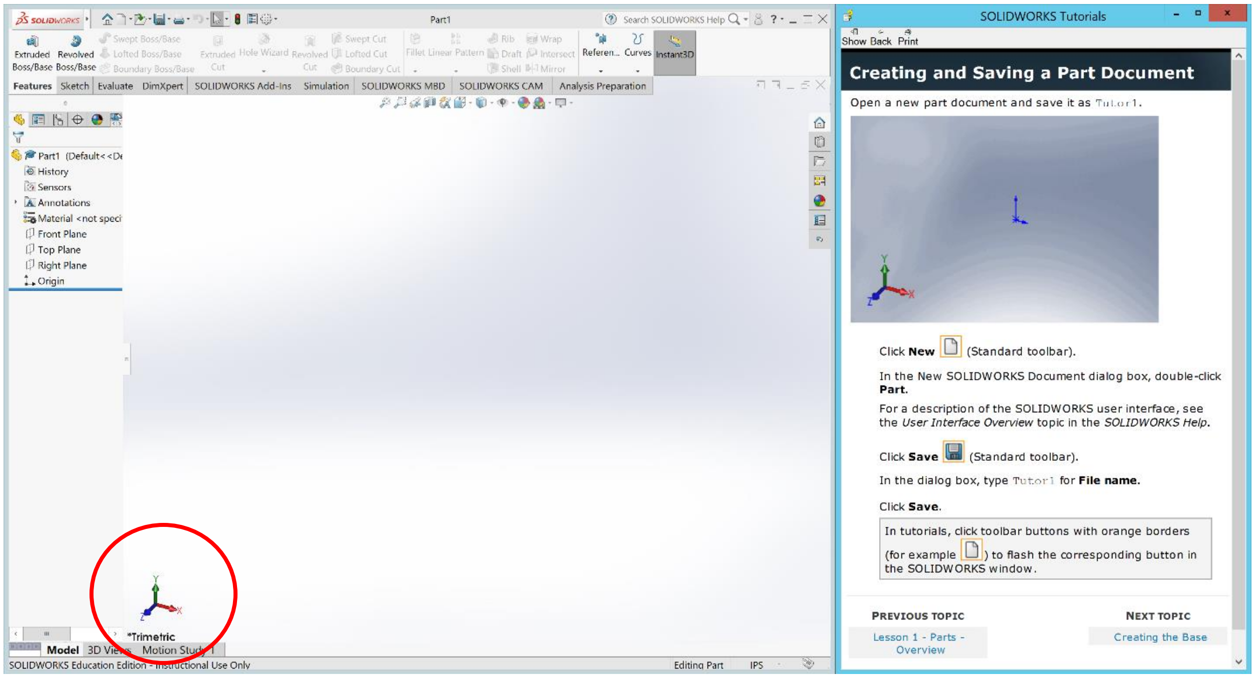
A menu appears, click on “Learn” on the top bar of the menu as circled in red below, then click on the middle blue square “Tutorials”:





Click on "Lesson 1: Parts" as seen in the image to the right. Follow along with the detailed instructions provided by SOLIDWORKS to create your first part. Make sure you save your part on your Z: drive ***often***! Once you have clicked through the instructions to create a new "Part", your screen should look like the image below. If you **have trouble finding any of the tools** used in the tutorial, click on the icon in the tutorial itself and it will indicate the icon location on the actual SOLIDWORKS menu for you by flashing the icon.

What do the coloured arrows (circled in red in the image below) represent?



Creating and Saving a Part Document

Open a new part document and save it as Tutor1.

Click **New** (Standard toolbar).

In the New SOLIDWORKS Document dialog box, double-click **Part.**

For a description of the SOLIDWORKS user interface, see the *User Interface Overview* topic in the *SOLIDWORKS Help*.

Click **Save** (Standard toolbar).

In the dialog box, type Tutor1 for **File name.**

Click **Save**.

In tutorials, click toolbar buttons with orange borders (for example) to flash the corresponding button in the SOLIDWORKS window.

PREVIOUS TOPIC
Lesson 1 - Parts - Overview

NEXT TOPIC
Creating the Base

Once you have completed “Lesson 1: Parts”, make sure you have properly saved your part on your Z: drive. You must also save your part as an image which you will upload to Virtual Campus as part of your lab assignment. To save your part as an image, click on the little down arrow next to the save icon in the menu at the top of the window . Click on “Save As”. In the window that opens up, click on the little down arrows for the “Save as type” menu and select the PDF file format and save your part again on your Z: drive.

Saving in a File Format Compatible with 3D Printing

If you intend to 3D print your part, you must save your part using a “.stl” file format. To do so, click on “Save As”. In the window that opens up, click on the down arrow for the “Save as type” menu and select the STL file format and save your part again on your Z: drive. The part file saved in the STL file format can be imported directly into any of the 3D printing software used at the Makerspace.

Tutorial 2: Creating an assembly

You will now make an assembly using a SOLIDWORKS tutorial. Once again, click on the “SOLIDWORKS Resources” icon on the right-hand side of the main window, click “Welcome to SOLIDWORKS”. Click the “Learn” tab, select “Tutorials” and click on “Lesson 2: Assemblies” as seen in the image to the right. Follow along with the detailed instructions provided by SOLIDWORKS to create your first assembly. Make sure you save your assembly on your Z: drive ***often***!

SOLIDWORKS lets you calculate analyze and simulate many things. For example, using the “Evaluate” tab in the “CommandManager” toolbar, as shown in the image below, you can select the “Mass Properties” command to give you all kinds of information about your assembly. What is the volume of your assembly?

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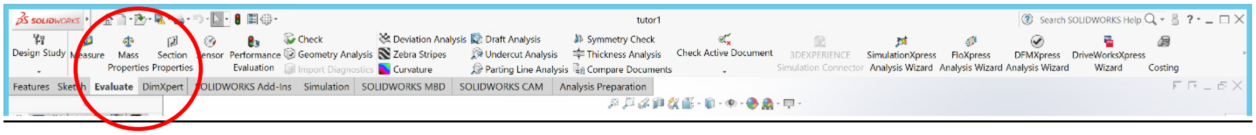
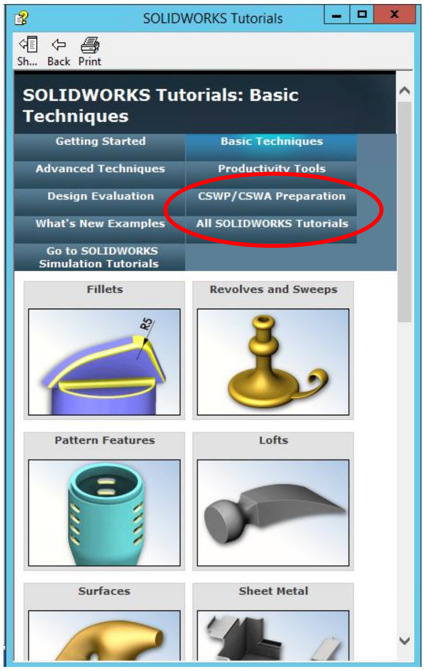


Once you have completed “Lesson 2: Assemblies”, make sure you have properly saved your part on your Z: drive. You must also save your assembly as an image which you will upload to Virtual Campus as part of your lab assignment. To save your assembly as an image, click on the little down arrow next to the save icon in the menu at the top of the window . Click on “Save As”. In the window that opens up, click on the down arrow for the “Save as type” menu and select the PDF file format and save you assembly again on your Z: drive.

Tutorial 3: Creating a personalised part (Explore)

As part of the lab assignment, you will need to provide one more PDF image of a 3D model you have created, this is your “personalized part”. Feel free to explore the various features in the SOLIDWORKS “CommandManager” toolbar when creating your model. You can also explore more techniques by following other tutorials, such as those in the “Basic Techniques” shown in the image to the right. Particularly useful techniques to learn include “Revolves and Sweeps”. Each student must create their own personalized part, using any of the SOLIDWORKS features (revolve, loft, helix, etc).



Submission

Submit this lab manual as a ***printed PDF*** with answers provided to the questions on the lines provided by uploading it to Virtual Campus before the due date, along with three PDF images of the “part”, “assembly” and “personalized part” and one STL file of the “part” you created during this lab. All uploaded components should be submitted as a single submission with multiple uploads, rather than separate submissions.

3D Printing of a Part

If time permits and you are interested, you can print the “part” you created in tutorial 1 on one of the 3D printers found in the lab. The following instructions will help guide you.

Before starting a print, students should become familiar with the printer they will be using, and be able to identify and properly use all of the main components.

Printer components

User Interface

On the front of the Ultimaker 2+ there is an SD card slot, a display screen, and a dial used for navigation, whose centre is also a button. On the back of the printer’s chassis is a power switch, a USB connection port, and the power cable.

Extruder and Nozzle (CAUTION: HOT!)

The extruder heats and pulls partially melted filament into the nozzle. During a print, the extruder and nozzle will heat up to over 200ºC, so exercise caution around it. The location of the printer nozzle and extruder is controlled on an axis system made up of belts and gears. This assembly can be moved while the printer is idle by gently pulling on the extruder/nozzle assembly. Be careful as parts of this assembly can be extremely hot even after a print has finished. If the printer is printing, or has recently been printing the motors will still be engaged. Set the printer to idle and wait a few minutes, or power off the machine to disengage the motor lock.

Build Plate (CAUTION: HOT!)

The build surface is the surface on which the printed part built. On the lab printers, this is heated to 110ºC during printing, so exercise caution around it. The plate can be raised or lowered while the printer is idle by going to Maintenance>Advanced>Raise/Lower Build Plate. If the printer is not printing properly, this may need to be adjusted by using the three knobs on the underside of the plate. This can be leveled through the printer's onboard software by going to Maintenance > Build Plate, and following the onscreen instructions.

Filament Spool

The filament spool can be found attached to the back of the printer. If the printer is out of filament, it can be replaced by going to Material>Change in the printer's user interface and following the onscreen instructions. It can also be replaced midway through printing by selecting Pause>Change Material and following the onscreen instructions.

Setting up the print

Students must save their SOLIDWORKS 3D model as an .stl file. Moreover, if a student wishes to use their own computer instead of the lab computers for printing, they may do so, provided that Cura 3.4.1 (or the latest version) is downloaded and installed beforehand.

Setting Up Cura 3.4.1

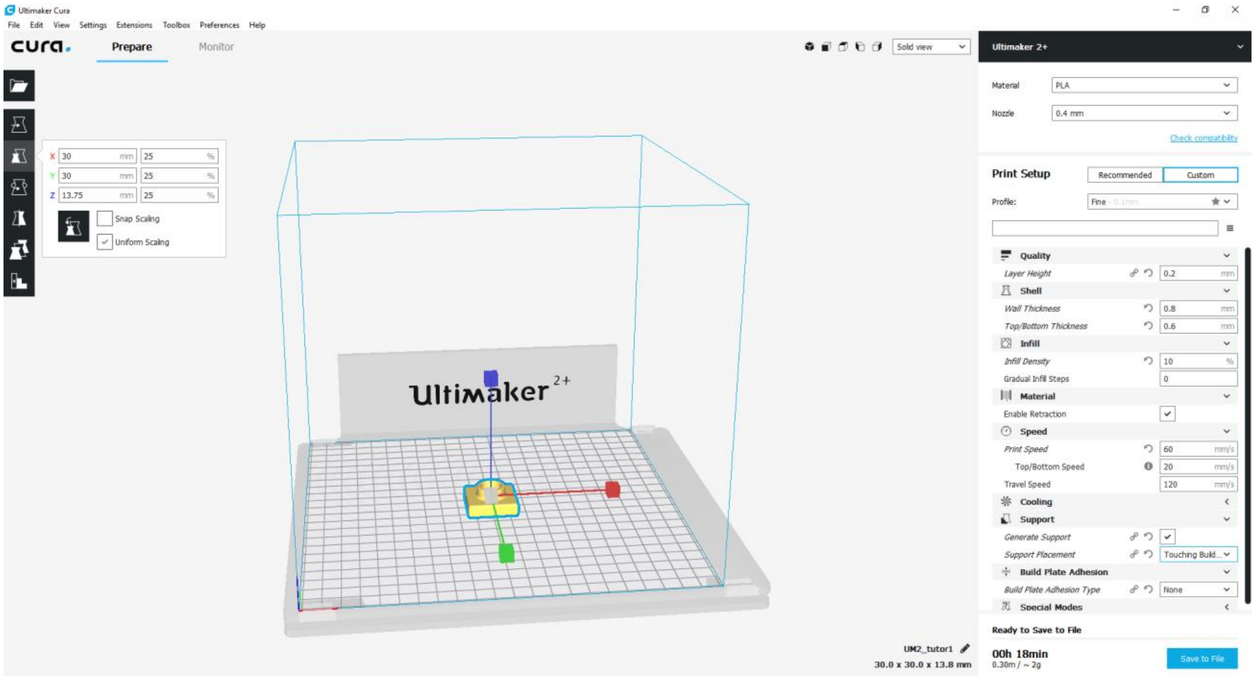
To install the most recent version of the Cura software used by the Ultimaker 2+ go to:

https://ultimaker.com/en/products/cura-software

and download the software (default is for 64-bit versions of windows). Settings may have to be adjusted to fit the recommended settings used in this lab.

Procedure

1. Open Cura 3.4.1 and make sure the necessary .stl files have been saved.
2. In Cura's main window, select "Load" and select a .stl file to open. The part will load in the middle of the virtual build plate.
3. Select the part by clicking on it, then select the scale icon in the top left-hand corner. Change the 'Scale X' number to 0.25 for 25%. This will change all dimensions proportionally if 'Uniform scale' is locked.



X 30 mm 25 %
Y 30 mm 25 %
Z 13.75 mm 25 %
☐ Snap Scaling
☑ Uniform Scaling

Material: PLA
Nozzle: 0.4 mm

Print Setup: Custom
Profile: Fine - 0.1mm

Quality
Layer Height: 0.2 mm
Shell
Wall Thickness: 0.8 mm
Top/Bottom Thickness: 0.6 mm
Infill
Infill Density: 10 %
Gradual Infill Steps: 0
Material
Enable Retraction: ☑
Speed
Print Speed: 60 mm/s
Top/Bottom Speed: 20 mm/s
Travel Speed: 120 mm/s
Cooling
Support
Generate Support: ☑
Support Placement: Touching Build...
Build Plate Adhesion
Build Plate Adhesion Type: None
Special Modes

UM2_tutor1
30.0 x 30.0 x 13.8 mm

Ready to Save to File
00h 18min
0.30m / ~ 2g

4. Ensure that the print settings have been properly configured by referencing the table below. You will need supports since this is a hollow piece. Verify the estimated print time matches the print time in the table for the part being printed. This will ensure that the part will print quickly while still maintaining accuracy.

Part: Tutor 1, Estimated print time (min) = 18	
Basic Settings	
Nozzle Size (mm)	0.4
Layer Height (mm)	0.2
Shell Thickness (mm)	0.8
Bottom/Top Thickness (mm)	0.6
Fill Density (%)	10
Retraction Enabled	Yes
Print Speed (mm/s)	60
Support Type	Touching Builplate
Build Plate Adhesion Type	Brim

5. Insert an SD card into the computer or connected card reader and go to File>Print. This saves a .gcode file to an external storage device. If there is more than one device connected (for example, an SD card and USB flash drive), Cura will ask which one to use.
6. Safely eject the SD card with the .gcode file on it from the computer and insert it into the Ultimaker 2+ printer.
7. Before starting the print, ensure the printer:
    - Has enough filament for the print (found on the spool on the back of the printer)
    - Nozzle and build plate are clear of debris

7. Replace filament spool and clean debris away if necessary. Heating the build plate or nozzle may help in removing plastic (CAUTION: use a towel or pliers to avoid risk of burns from the nozzle or build plate).
8. Select Print>yourfilename.gcode in the printer's user interface. The printer will take a few minutes to heat up, then begin to print.
9. Monitor the print job for major errors in extrusion or adhesion of the plastic. If errors do occur in the print, it can be paused or aborted (Tune>Abort) using the printer's interface.

*NOTE: The printer gives a readout estimating the remaining print time, and during the first few layers this estimate is usually a gross overestimate.

After the print has completed, allow the build plate to cool for a few minutes before removing the print. Trim any excess plastic from the component (including support structures), and ensure the printer is clear of debris and ready for the next print.