Lecture 1 - Setting your workspace

Setting your workspace is very important before you do any work on SOLIDWORKS. It can save you a lot of time during the CSWA and CSWP exams if the computer that you will use is slow.

1 Launching SOLIDWORKS

At first, you will need to open the SOLIDWORKS software. To do so, click on .

This icon can be found either in your desktop or you can search for SOLIDWORKS in the search bar of your Windows desktop or laptop. The search bar mentioned earlier is seen in Figure 1 and the result of your research is shown in Figure 2

Figure 1: Search Bar in Windows 10

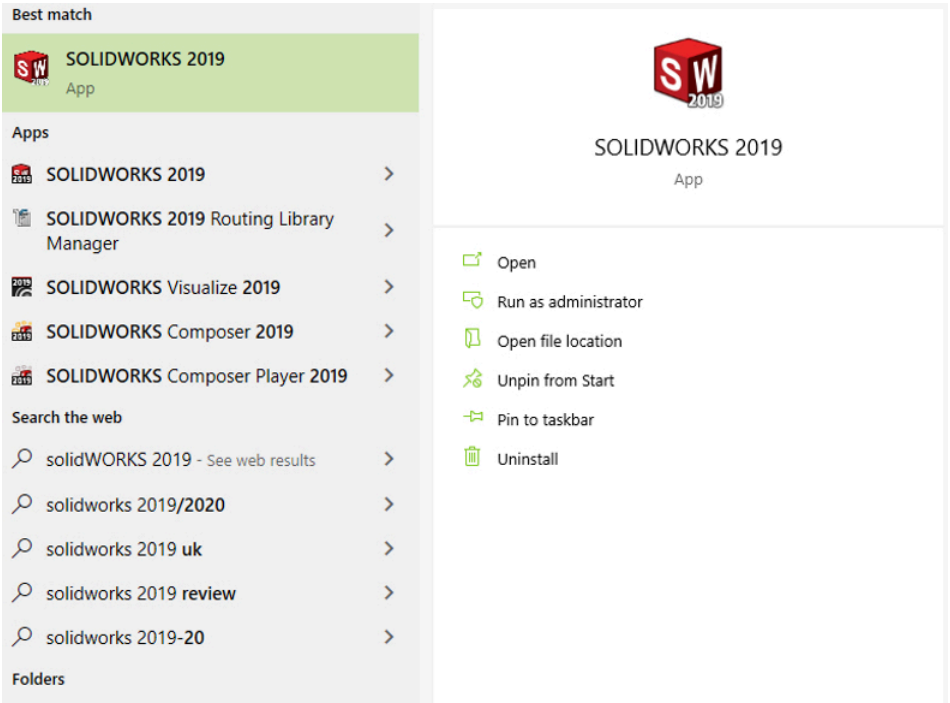


Figure 2: Result of your research in the Search Bar

2 Selecting the SOLIDWORKS document type

Once your software is opened, you will the window seen in Figure 3 on your screen

Figure 3: SOLIDWORKS Welcome Window

There are two ways to create a new SOLIDWORKS file from the welcome window:

– You can create such file by clicking **Home**, which can be seen in the toolbar at the top of your screen. Once done so, you will obtain the following window.

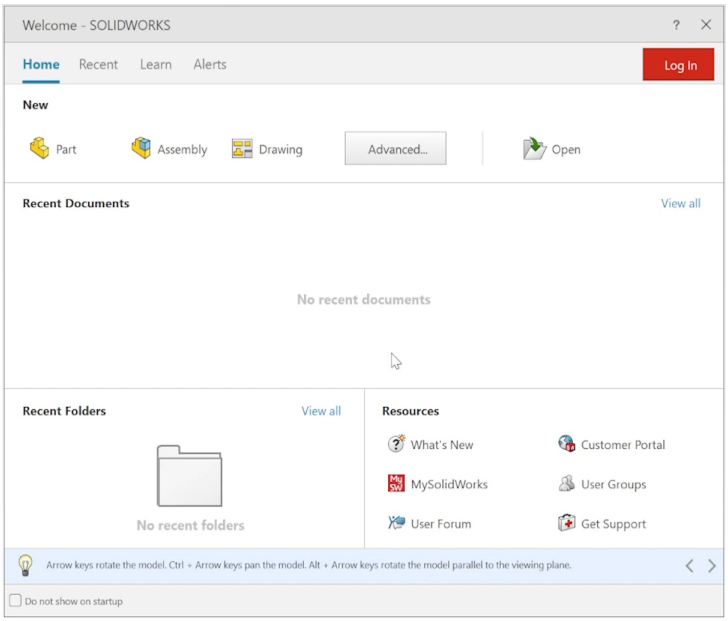


Figure 4: SOLIDWORKS document type selection window

– You can also create such file by clicking **New**, which is located next to the **Home** button mentioned above. Once done so, you will obtain the following window.

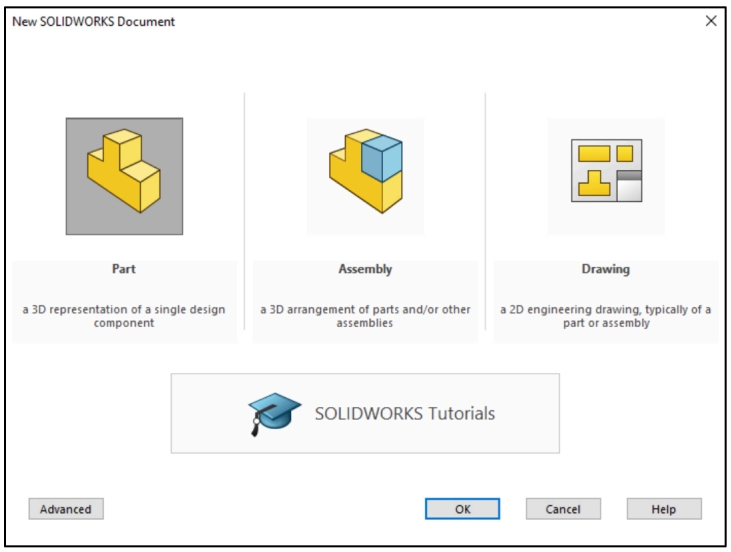


Figure 5: SOLIDWORKS document type selection window

The window seen in Figures 4 and 5 allows you to select what do you want to create. As you can see in Figures 4 and 5, there are three options to select from:

– A **part** is a three-dimensional object that contains features, in order words shapes.

– An **assembly** is a combination of multiple parts. Parts must be created beforehand before creating an assembly.

– A **drawing** is a two-dimensional representation of your three-dimensional part and/or your assembly that you created beforehand. In a drawing, all part dimensions must be specified.

For learning the essential skills in SOLIDWORKS, we will be designing a simple mug, which can be seen in Figure 6.

Figure 6: Mug to be designed between Lectures 1 to 4

Since the mug is a part on its own, click on **Part** to create a part in SOLIDWORKS. The following window will appear.

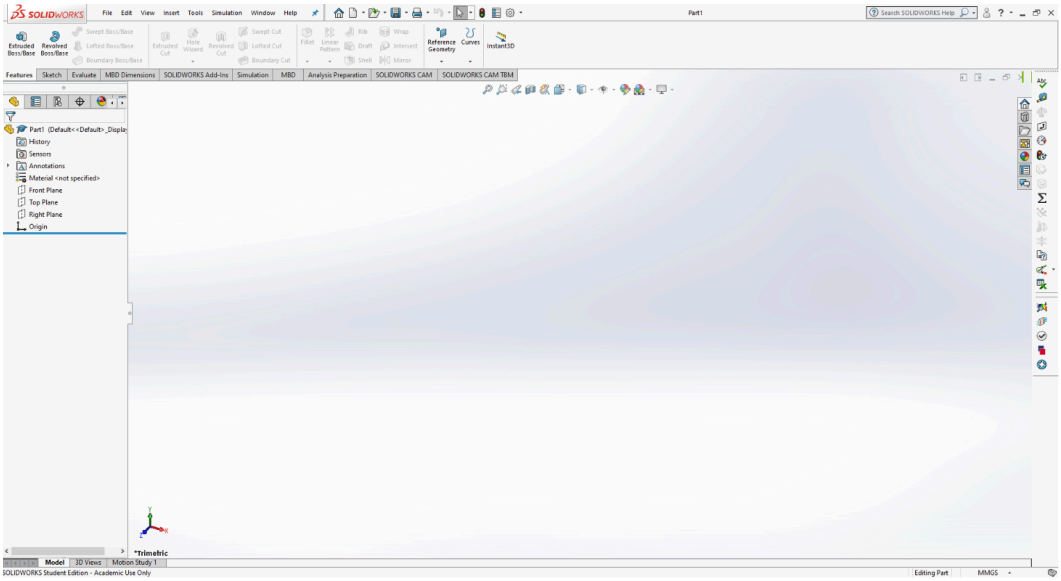


Figure 7: Sketch Editor

The window displayed in Figure 7 is the **Sketch Editor**. Do not be alarmed by the complexity of this window when you look at it for the first time. You will learn more about the Sketch Editor in Lecture 2 onward.

3 Units and dimension standards in SOLIDWORKS

3.1 Units

Before you start to do your work, you need to check the units that you are using in this document. It is important as a designer and as an engineer to be coherent with your units. There are four systems of units that you can choose from:

- Metre Kilogram Second or **MKS**
- Centimetre Gram Second or **CGS**
- Millimetre Gram Second or **MMGS**
- Inch Pound Second or **IPS**

The **IPS** system is known as the imperial unit system. This system of units is not that much used in engineering in most countries, it is still being used in the United States. For avoiding confusion, such system of units will not be used in this course.

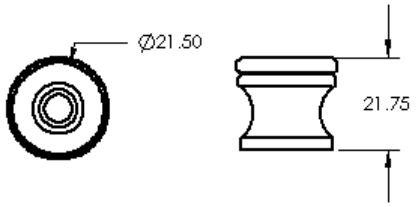
The **MKS** system is known as the metric or SI units, which is the 'international standard of measurements" (NIST [no date]), represented by the following base units:

- Length, metre (m)
- Time, second (s)
- Amount of substance, mole (mole)
- Electric current, ampere (A)
- Temperature, Kelvin (°K)
- Luminous intensity, candela (cd)
- Mass, kilogram (kg)

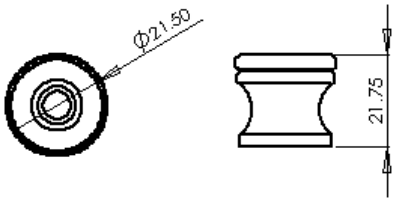
The **MMGS** and **CGS** systems of units are both derived from the MKS system described previously. **For CAD designing, it is recommended to work with the MMGS system of units** as most measurements, often made with callipers, are made in millimetre. **Therefore, for the series of lectures, the MMGS system of units will be used.**

3.2 Dimension standards

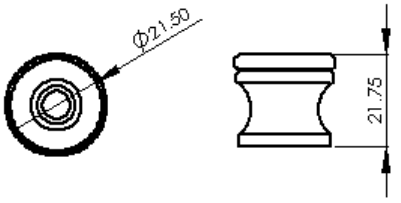
In SOLIDWORKS, there are different ways to format part dimensions in a drawing. These depend on the country in which your part is designed. They can be seen in Figure 8. For this series of tutorials, the dimension standards used is ANSI, which is illustrated in Figure 8a.



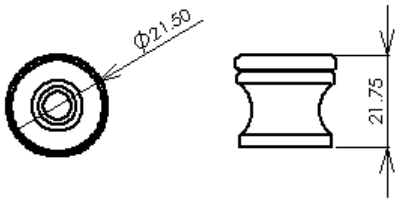
(a) ANSI - American National Standard Institute (USA)



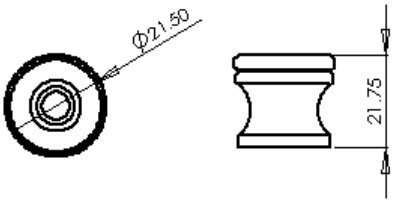
(b) ISO - International Organisation for Standardisation



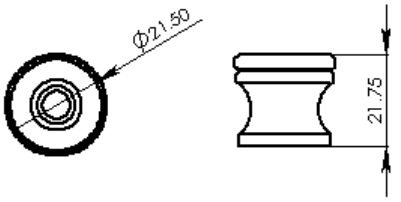
(c) DIN - Deutsches Institut für Normung (Germany)



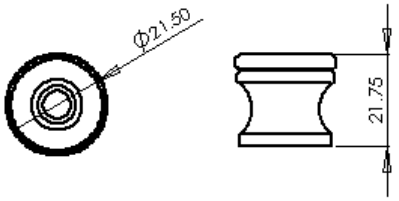
(d) JIS - Japanese Industrial Standards (Japan)



(e) BSI - British Standards Institute (UK)



(f) GOST (Russia)



(g) GB - National Standards of the People's Republic of China (China)

Figure 8: Dimensions standards available in SOLIDWORKS, figure reproduced from [1]

3.3 Settings the units and dimension standards used in SOLIDWORKS

3.3.1 Set your units and dimension standards after a clean install

If it is the first time you launch SOLIDWORKS, you have a window that will automatically request you to set up the units and the dimension standards. Such window can be seen in Figure 9.
As discussed in sections 3.1 and 3.2, we desire to work with the ”MMGS” units and ”ANSI” dimension standards. To set these,

- Click on the ”Units” selection wheel and select ”MMGS”.
- Click on the ”Dimension Standards” selection wheel and select ”ANSI”.

To validate your selections, please click on ”OK” to continue to the next step seen in Figure 7.

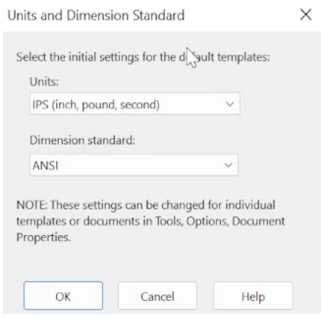


Figure 9: Units and Dimension Standard selection window

3.3.2 Change units in SOLIDWORKS

If your laptop or desktop has SOLIDWORKS pre-installed and wish to change the units and dimension standards, you can easily change them by following these steps. Figure 10 (see next page) illustrates where to check which system of units is being used in the project. If the MMGS system of units is not selected, please select 'MMGS' from the unit selection wheel that can be seen in Figure 11. You can also change the units by going to **Tools > Options > Document Properties > Units**. You will then obtain the window displayed in Figure 12.

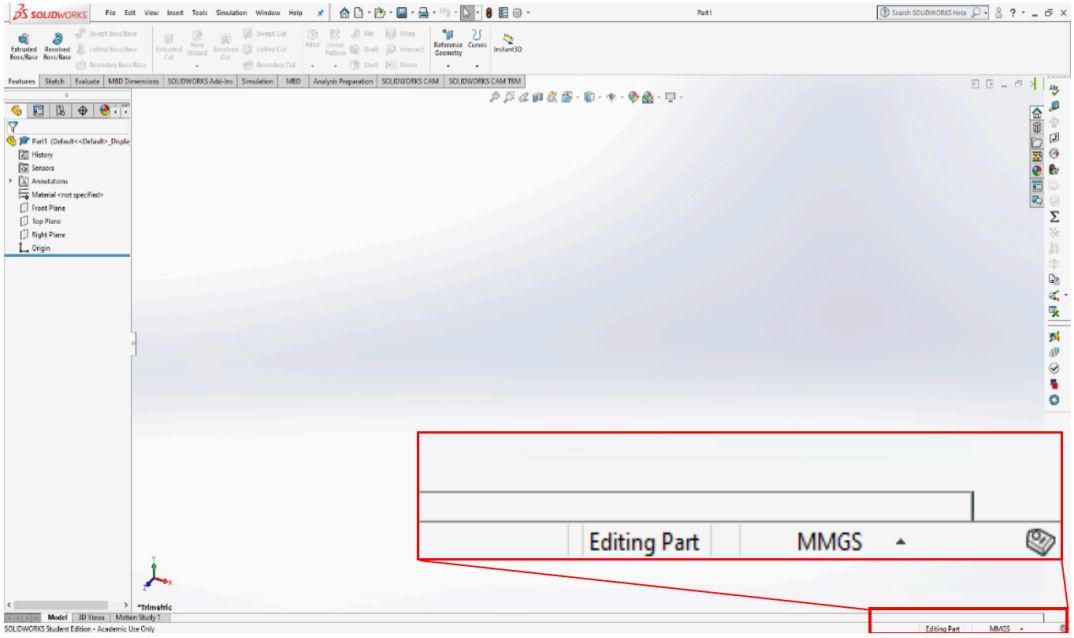


Figure 10: Location of system of units used in the Sketch Editor

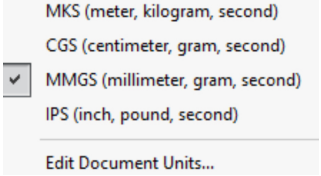


Figure 11: Unit Selection Wheel

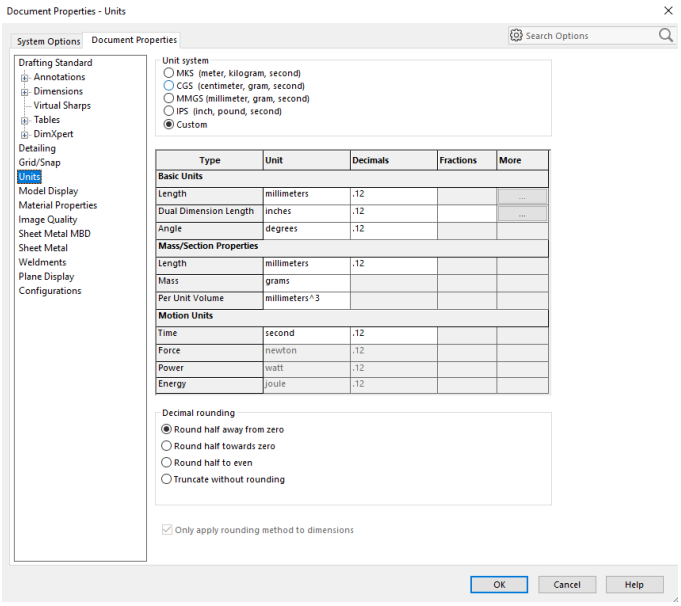


Figure 12: Output of the path **Tools > Options > Document Properties > Units**.

3.3.3 Change dimension standards in SOLIDWORKS

For changing the dimension standards in SOLIDWORKS for your drawings, go to **Tools > Options > Document Properties > Drafting Standard**. After following this path, you will obtain the window seen in Figure 13.

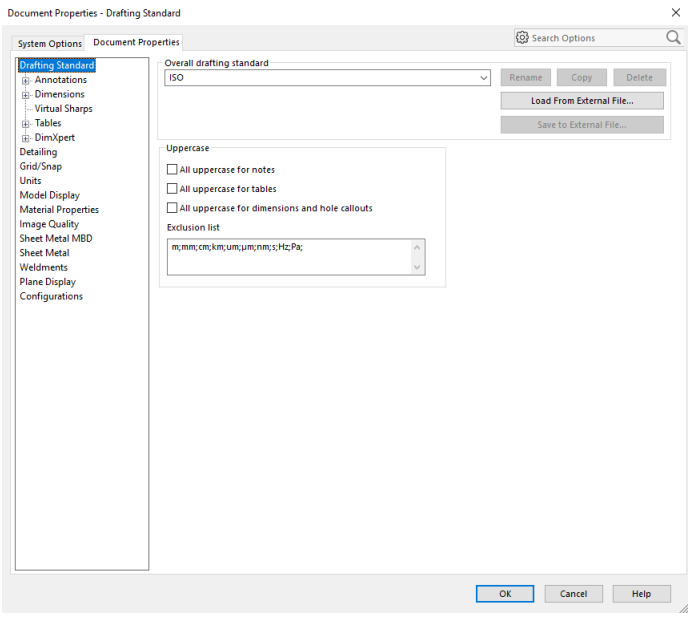


Figure 13: Output of the path **Tools > Options > Document Properties > Drafting Standard**.

Once you have defined the units that you will be working with, you are now ready to create your part in SOLIDWORKS.

NEXT LECTURE: A series of four lecture videos will guide you in the creation of the mug's main body via extrusion. The entire process will be summarised in Lecture 2.

References

[1] D. Systemes, “Standards - 2020 - SOLIDWORKS Help.” [Online]. Available: https://help.solidworks.com/2020/english/SolidWorks/acadhelp/c_standards_acadhelp.htm