# 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

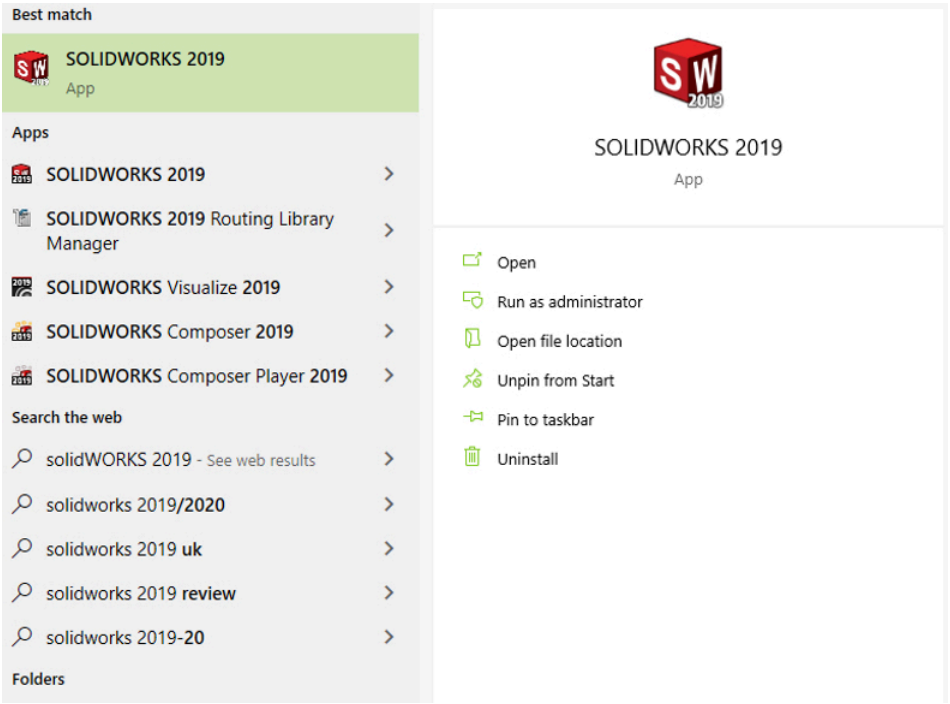


Figure 2: Result of your research in the Search Bar

## 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

Now, to create a new SOLIDWORKS file you will need to click **New** . Once done so, you will obtain the following window.

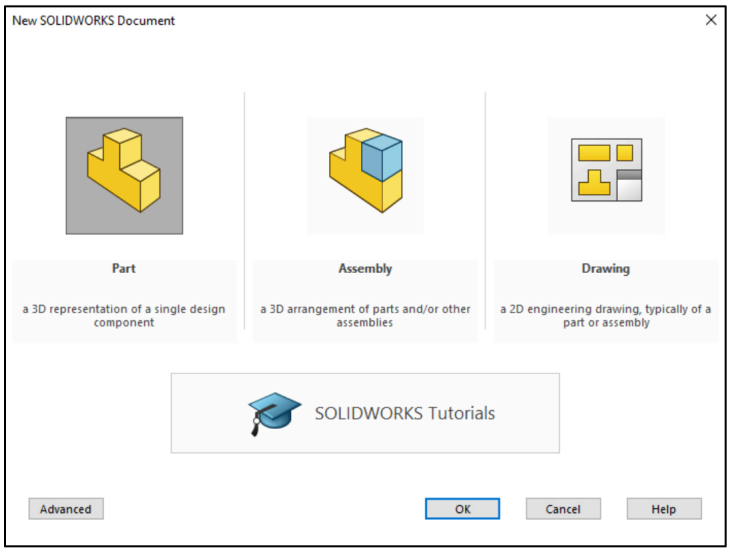


Figure 4: SOLIDWORKS document type selection window

The window seen in Figure 4 allows you to select what do you want to create. As you can see in Figure 4, 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 the following drawing.

Figure 5: 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.

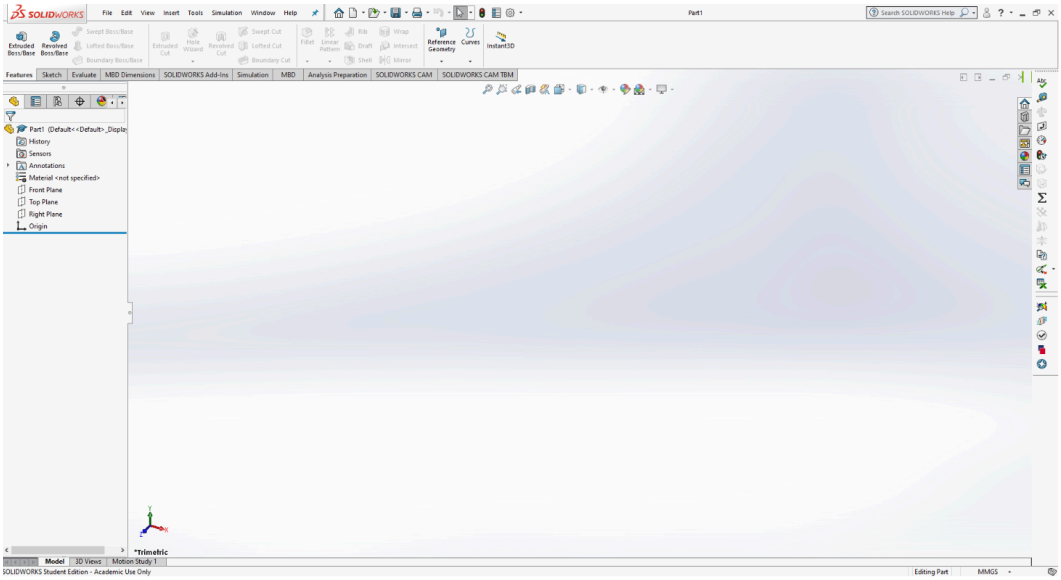

Figure 6: Sketch Editor

The window displayed in Figure 6 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.

# 3 Setting the 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.**

### 3.1.1 Settings your units after clean SOLIDWORKS install

### 3.1.2 Changing your units

Figure 7 (see next page) illustrates where to check which system of units is being used in the project.

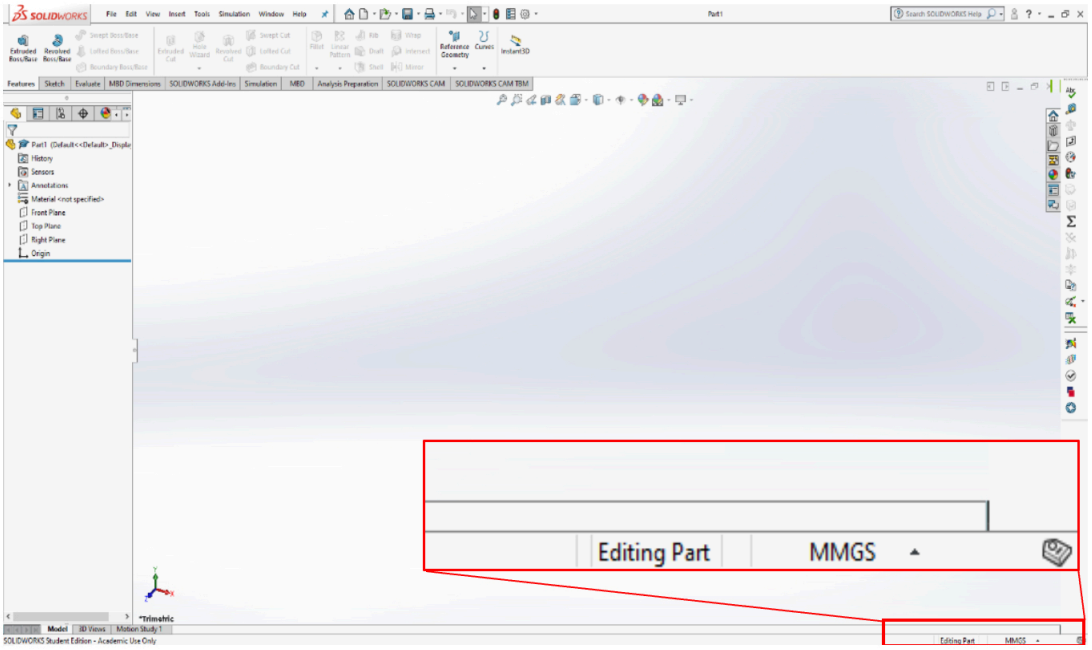


Figure 7: Location of system of units used in the Sketch Editor

If the MMGS system of units is not selected„ please select ‘MMGS’ from the unit selection wheel that can be seen in Figure 8.

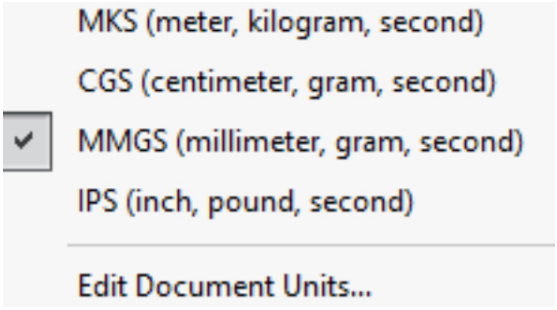


Figure 8: Unit Selection Wheel

### 3.2 Dimension Standards

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.