

Fakulti: FACULTY OF BIOSCIENCES AND MEDICAL ENGINEERING	
Nama Matapelajaran: Laboratory 1 Kod Matapelajaran : SMBE 2712	Semakan : Tarikh Keluaran : 2013 Pindaan Terakhir : 2017 No. Prosedur :



FACULTY OF BIOSCIENCES AND MEDICAL ENGINEERING
UNIVERSITI TEKNOLOGI MALAYSIA

SMBE 2712 LABORATORY 1

STUDENT PACK

INTRODUCTION TO THE SOLIDWORKS™ DESIGN SOFTWARE

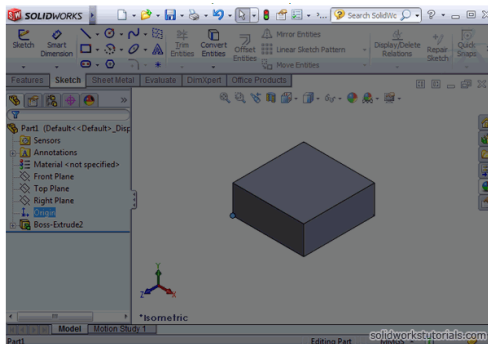
Disediakan oleh	Disahkan oleh
Nama : Arief Ruhullah bin A Harris	Nama :
Tandatangan :	Tandatangan :
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Tarikh : 10/9/2017	Tarikh :

Introduction

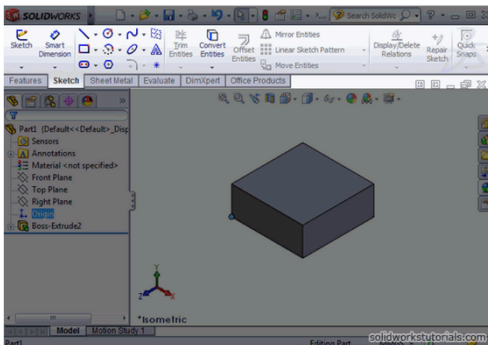
This laboratory consists of three sessions that requires the students to use the Solidworks™ software and complete Lesson based on tutorial. Upon complete both Solidworks laboratory session, students will able to design a simple biomedical application device: an external bone fixator. In the laboratory, students will be introduced to the basic technique for Solidwork, understand the concepts and the user interface available in the Solidworks™. Students will use the tutorial provided and follow the example accordingly for Lesson 1 and Lesson 2.

Solidworks concept is to allow user to create drawing directly in 3D or solid form. Solidworks offer user a simple and pretty straight forward User Interface. There is 6 main area of interface you normally work with. To learn more on Solidworks tutorial you can search for Solidwork Tutorial video using the internet search engine.

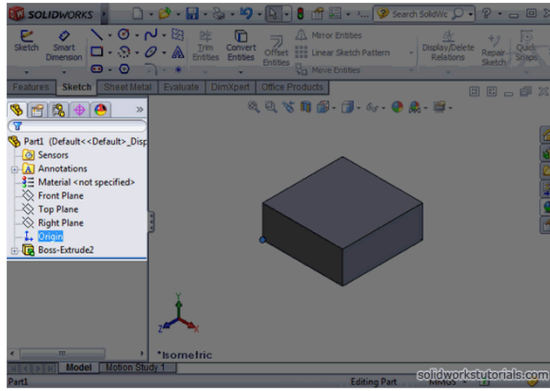
- 1) Menu Bar – Top most of the application, executing New File, Open File, Save, Print, Undo, Select, Rebuild, File Properties and Options.



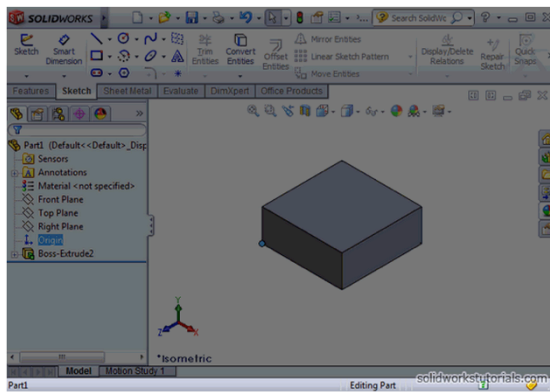
- 2) Command Manager – Access to part, assembly and drawing editing tools.



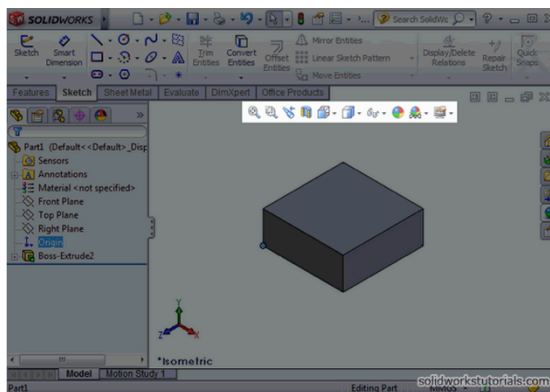
- 3) Feature Manager design tree – Outline overview how your part, assembly and drawing constructed.



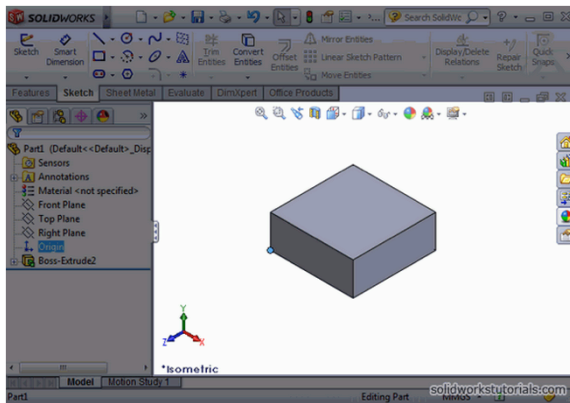
- 4) Status bar – Provide an information about your part, assembly and drawing.



- 5) Head up view toolbar – View tools such as zoom, pan, zoom plane and section view.

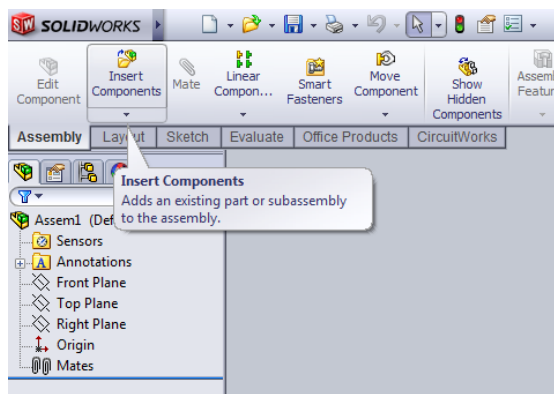


- 6) Graphics area – Workspace for your part, assembly and drawing.

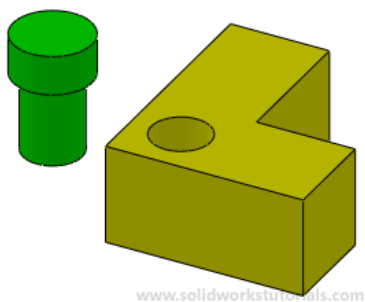
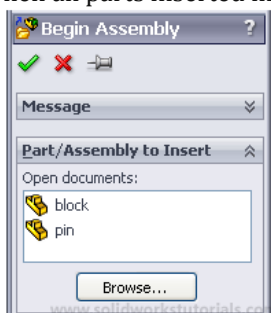


An assembly is a combination of two or more parts, also called components, within one SolidWorks document. You position and orient components using mates that form relations between components.

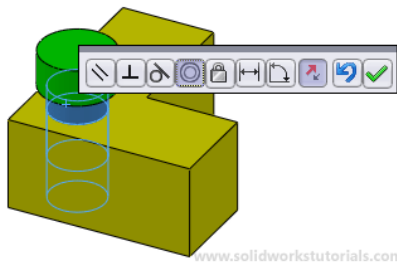
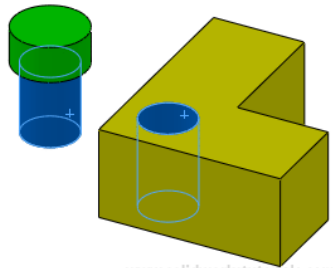
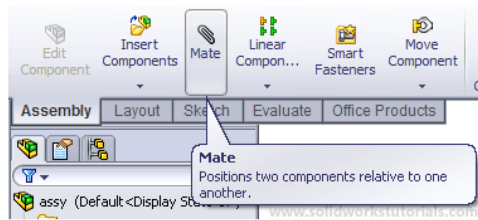
Assembly is how all parts work together in assembly, checking for clashes and its functionality. First all parts are inserted in assembly by the Insert Component tool.



When all parts are inserted into the workspace, the Mate command is used to define how parts mate with each other.



To combine the block and pin together: click Mate and select pin face and hole face, OK.



Objectives

1. To learn the basic concepts of Solidworks through the tutorial provided
2. To complete the tutorial in Lesson 1 and Lesson 2.

Equipments

1. Laptop
2. Solidworks™ installer

Pre-lab question:

1. What is the Solidworks™?
2. List any other software that can be used to design the 3D sketch and compare.
3. What is the Assembly function in Solidworks and why it is important?
4. List down any other functions/tools available in Solidworks.

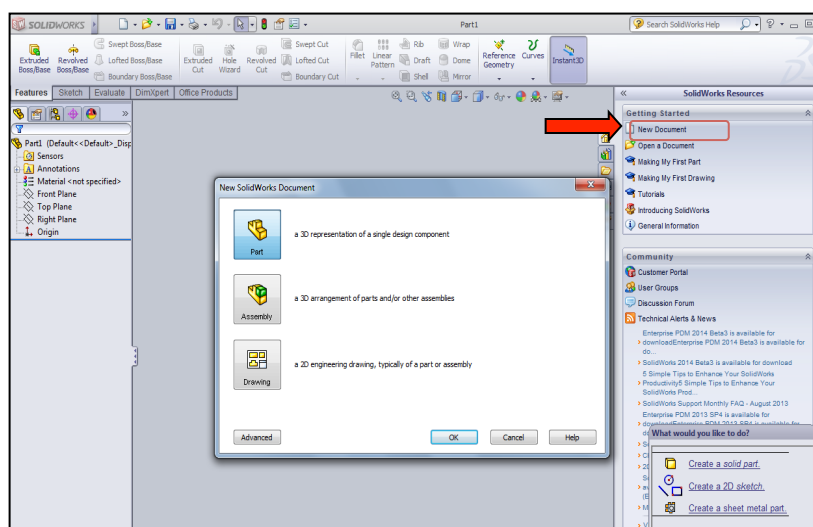
Experiment 1:

Requirement:

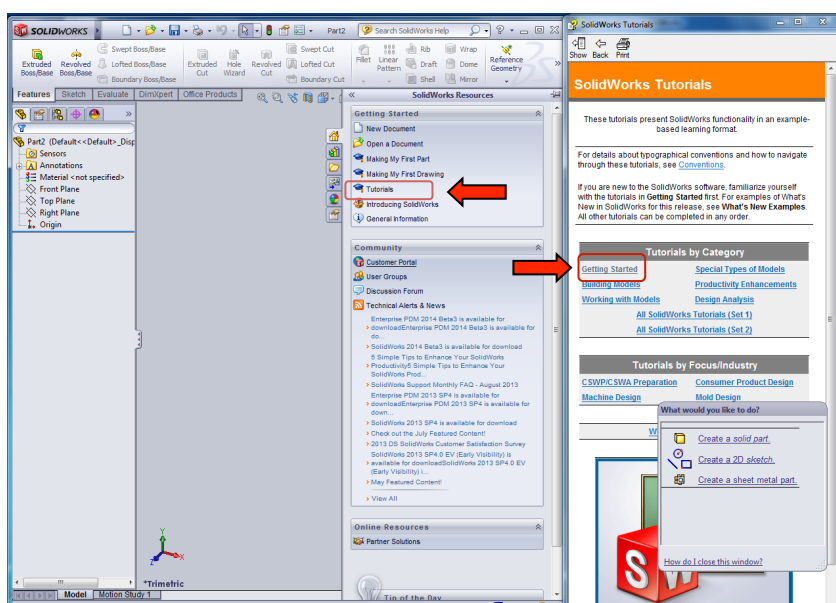
1. Students need to install the Solidworks™ software accordingly (32-bit/64-bit).
2. After the installation finish, double click the solidwork icon to start using it.

Instructions:

1. Once the SolidWorks window is open; in the right side of the windows click “New Document” and select the first option “a 3D representation of a single design component”



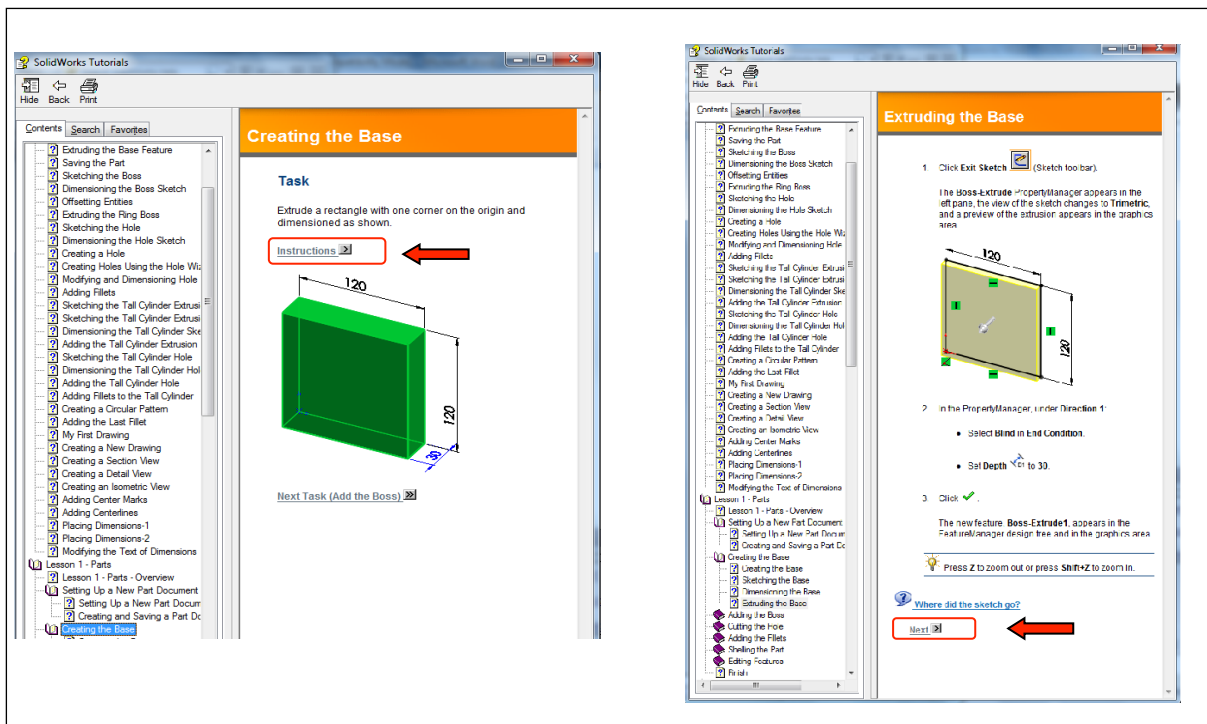
2. After that, click on “Tutorials” and the Solidworks tutorials window will popup. In the new window select “Getting started”.



- For this laboratory session, we will follow the Solidworks tutorial: Lesson 1. In the tutorials part, click "Lesson 1".

Overview on Lesson 1: In this lesson, you will create your first SolidWorks part.

- Setting up a new part document
- Creating the base feature
- Adding a boss feature
- Creating a cut feature
- Adding fillets
- Adding a shell feature
- Editing features



- Completed Part
- Click "Setting up the new document". Click the "instructions" and follow the step provided. You will be required to name the new document as "design1". After finish setup the document click "next".
 - You will follow all the step provided in the tutorials window and at each task follow the detailed instructions provided by click into "Instructions". Once finish each task just click on the "next".

Note: if you unable to locate the icon position, click on the icon provided in the tutorial and it will provide you the position.

- In the last step, the design is already done. Attach the result and provide the discussion on the techniques that you learn in the Solidworks design software.

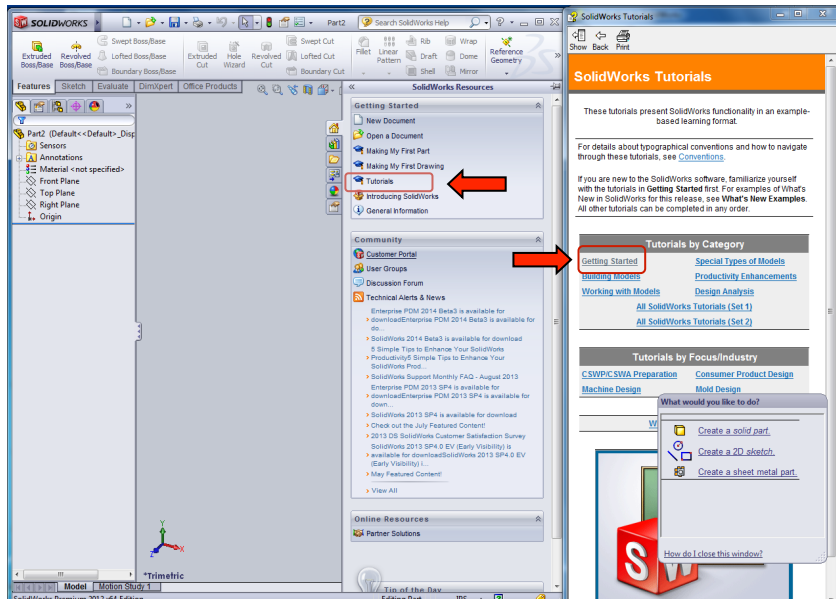
Experiment 2:

Requirement:

1. Students need to install the Solidworks™ software accordingly (32-bit/64-bit).
2. After the installation finish, double click the solidwork icon to start using it.

Instructions:

1. After the Solidworks windows open, click on “Tutorials” and the Solidworks tutorials window will popup. In the new window select “Getting started”.

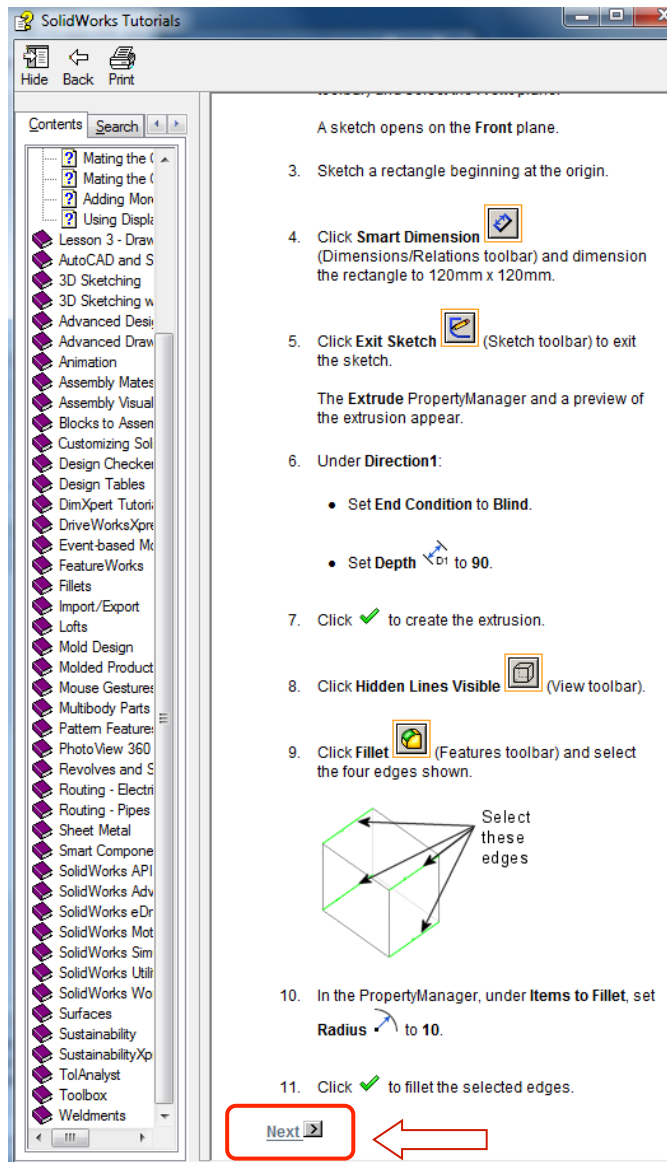


2. For this laboratory session, we will follow the Solidworks tutorial: Lesson 2. In the tutorials part, click “Lesson 2”.

In this lesson, you build a simple assembly based on the part you created in Lesson 1.

This lesson discusses the following:

- Adding parts to an assembly
 - Moving and rotating components in an assembly
 - Creating display states in an assembly
3. This lesson will be straight forward and you just need to click on “next”. You will be required to name the new document as “design2”. After finish setup the document click “next”. Follow the instructions provided and click next until you manage to assemble the first design done in the previous lab “design1” with the “design2”.



4. In the last step, the full design is already done. Attach the result on the space provided.

Result

Tasks	Lecturer/Officer initials
Experiment 1 done	
Experiment 2 done	

Discussion:**Conclusion****Lecturer approval**
