Hong Kong University of Science & Technology

Division of Integrative Systems and Design

ISDN 2400 Physical Prototyping

Lab 1 SolidWorks Introduction

Introduction

In this lab, we will learn the basic Computer Aided Design, CAD concept via SOLIDWORKS. SOLIDWORKS 2020 will be used in this tutorial. Part of the training material is based on the official training material which is available in https://www.SOLIDWORKS.com/. This tutorial is an intensive training such that you can get the basic technique to create 3D models quickly. Two of the key training materials, *INTRODUCING SOLIDWORKS* and *Student's Guide to Learning SolidWorks® Software (Student's Guide)*, are uploaded and will be used as reference.

Key Concepts

1. How a shape is constructed
2. The difference between a surface and a solid
3. What is a parametric design, and why do we need to make parametric design
4. Multiple parts and their relative positions
5. How to convert SolidWorks model for fabrication

Chapter 1 SOLIDWORKS Interface, Control and Basic Function

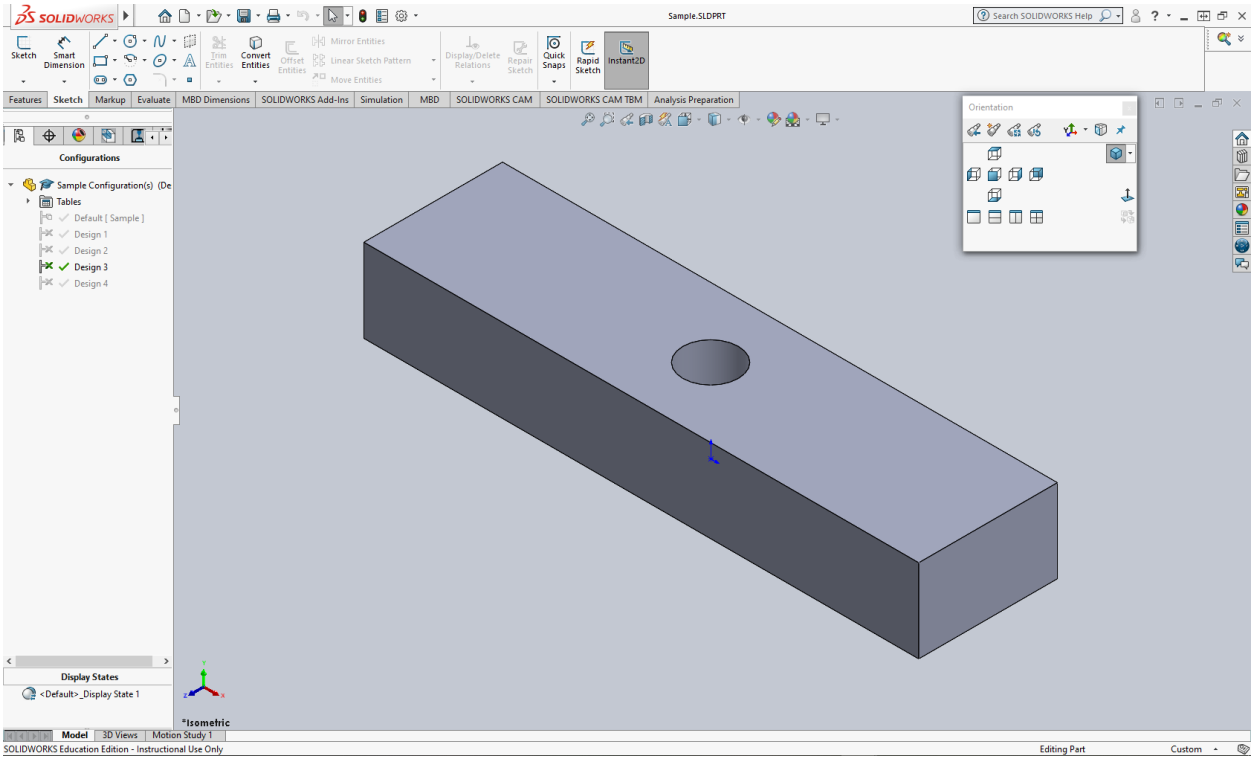
Objective

- Familiar with the SOLIDWORKS user interface
- Understand the basic control in SOLIDWORKS
- Use the basic functions to create simple geometry

Chapter 1.1 SOLIDWORKS Interface and Control

Reference

- *INTRODUCING SOLIDWORKS: P.11 – 23*



Chapter 1.2 Sketches

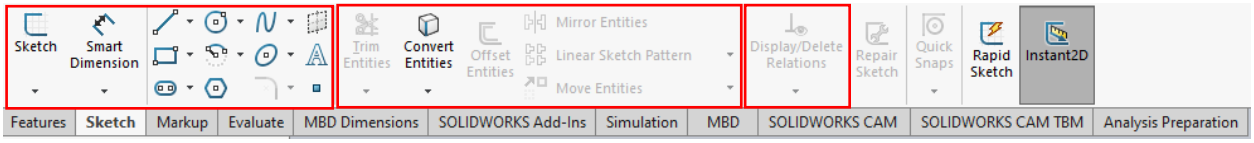
The sketch is the basis of most 3D modeling software including SOLIDWORKS. The key concept of sketched are:

- Origin
- Planes
- Dimension (Driving and Driven)
- Sketch Definitions (fully defined, under defined and over defined)
- Relation

Reference

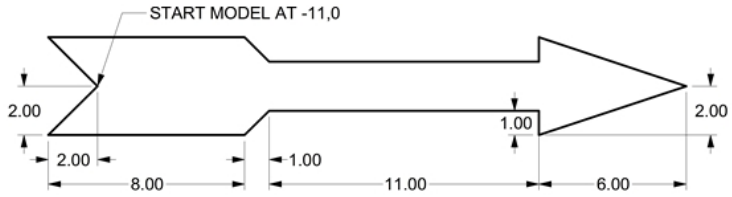
- *INTRODUCING SOLIDWORKS*: P.24 – 31
- *Student's Guide*: P.12 – 13

Sketch Tool

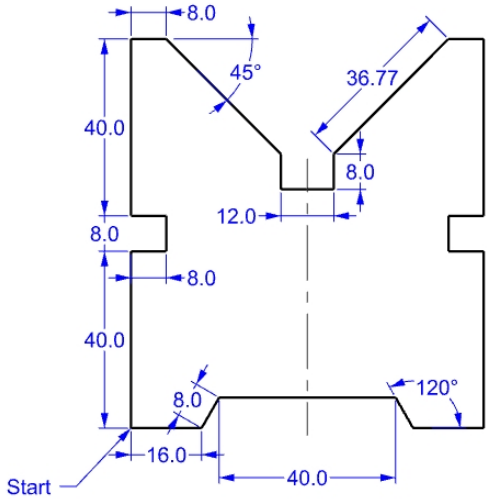


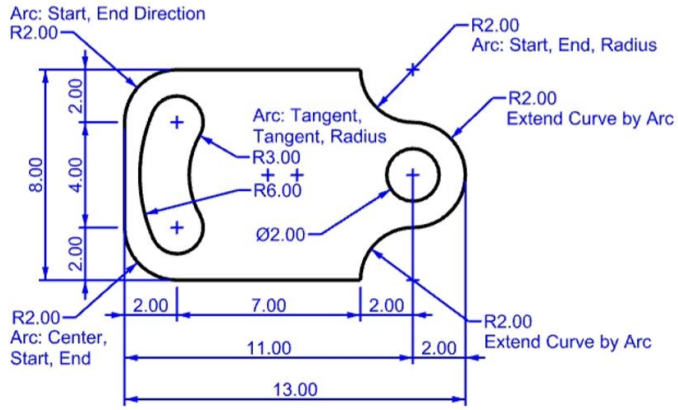
Example

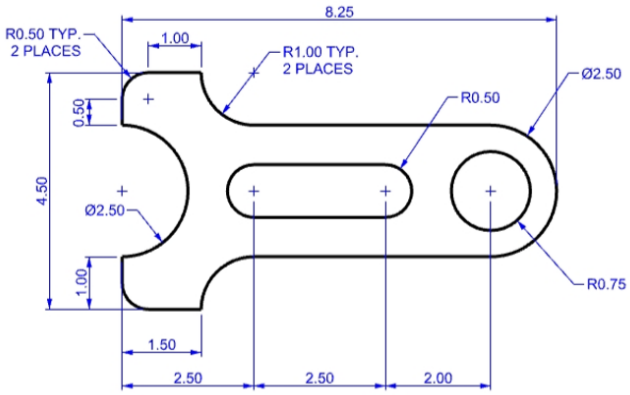
Arrow



Engineering Part







Chapter 1.3 Parts

After the creating the sketches, we can create solid based on the sketches. The key functions are:

- Extrude
- Extruded Cut

- Create Reference Plane

- Revolve
- Revolve Cut

- Sweep and Loft
- Sweep and Loft Cut

- Hole
- Geg
- Fillet and Chamfer

- Rapid Sketch (Create Plane and Sketch)

Reference

- *INTRODUCING SOLIDWORKS*: P.37 – 53

Example

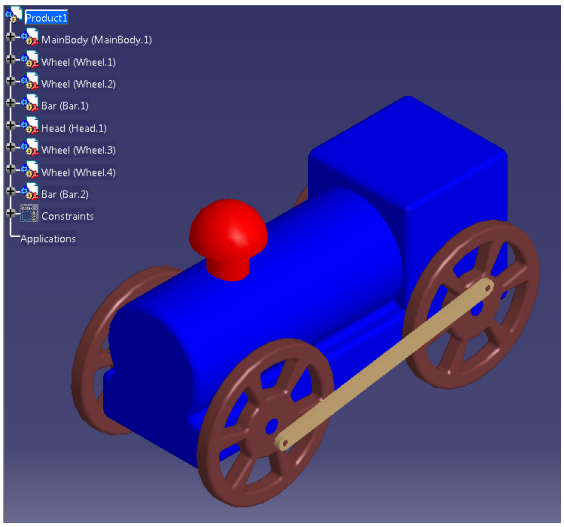
Chapter 1.4 More about Parts

In this session, we will look at more function in Parts which are:

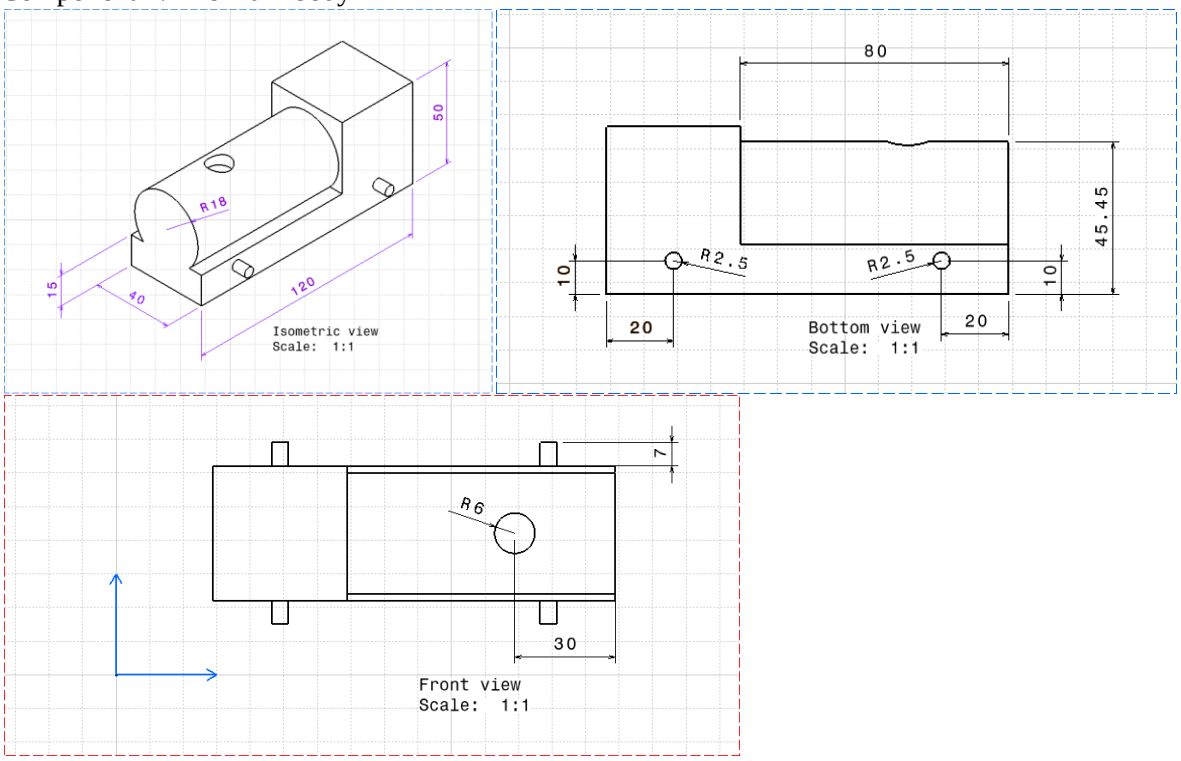
- Loft
- Fillet
- Pattern

Simple Toy Train Engine Example

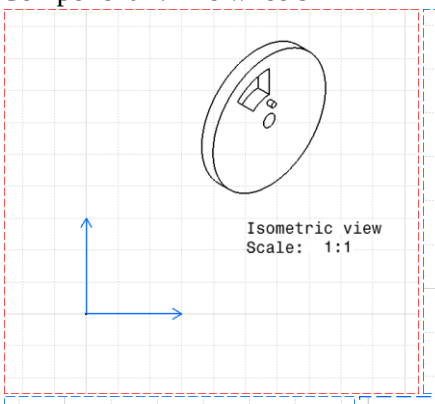
Draw the components of a toy train engine.

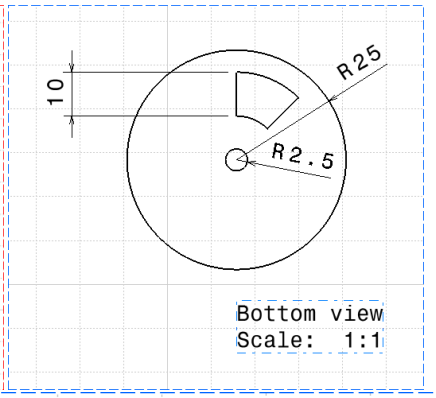


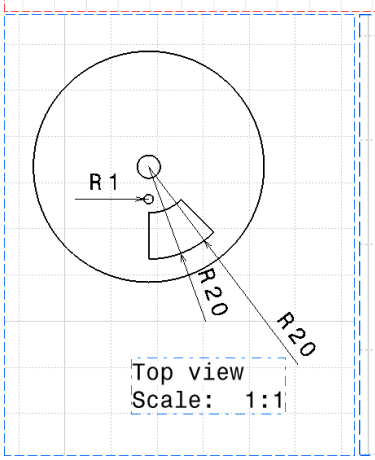
Component 1: The main body

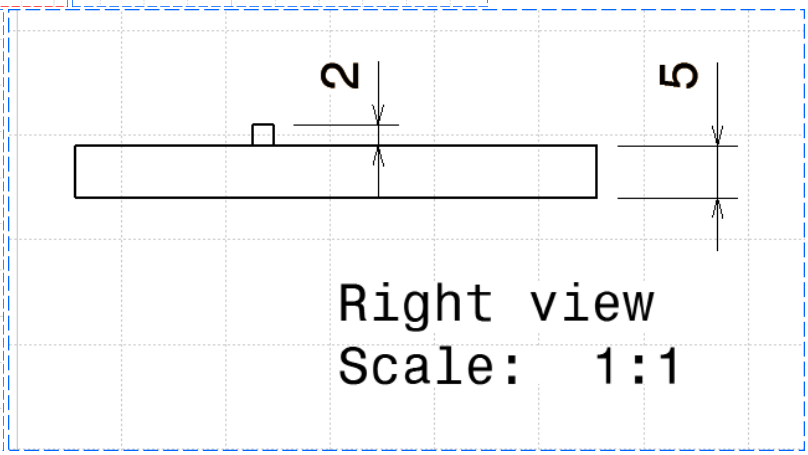


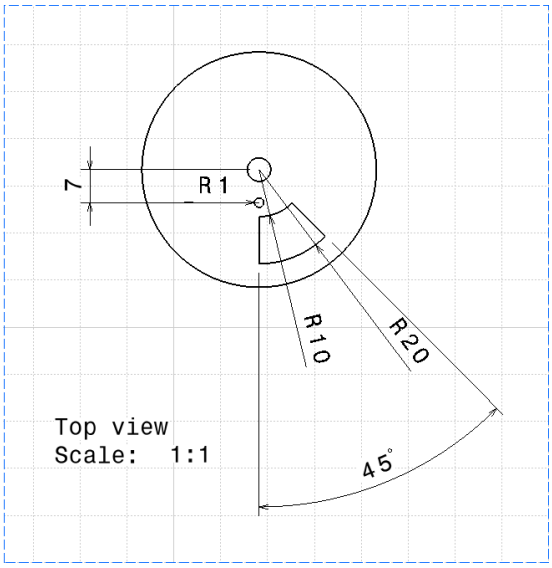
Component 2: The wheels



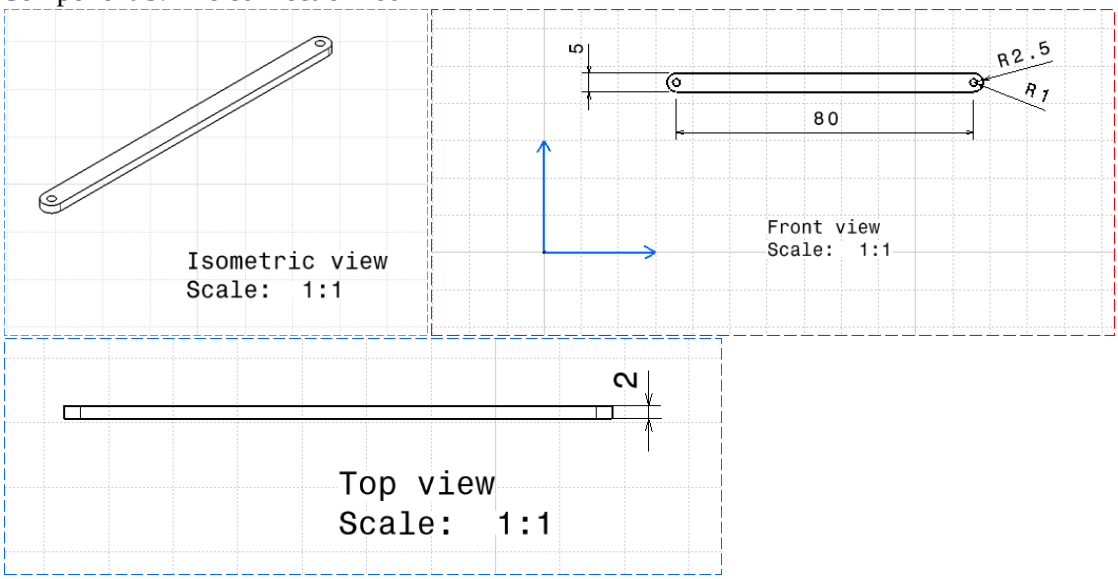




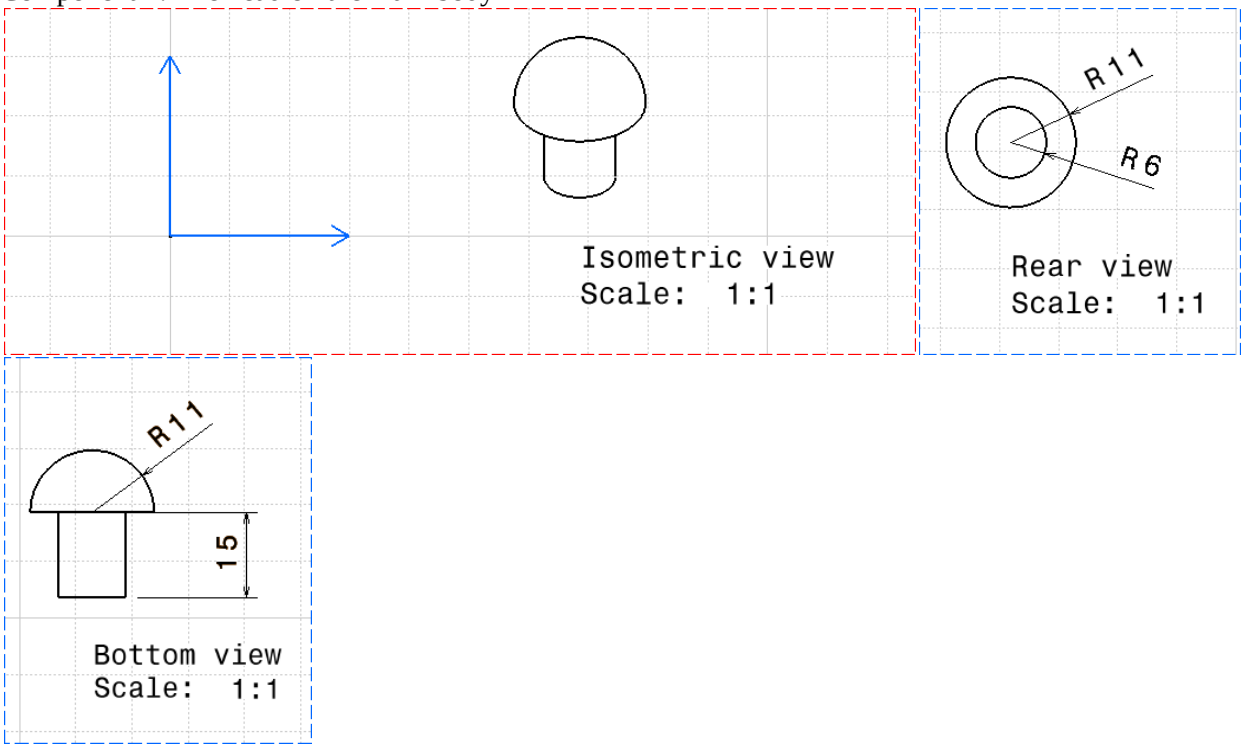




Component 3: The connection rod



Component 4: The head on the main body



Chapter 2 SOLIDWORKS Assembly

Objective

- Understand the basic concept of components in SOLIDWORKS Assembly
- Create relations between components
- Create animation or simulation of the assembly

Chapter 2.1 SOLIDWORKS Assembly Control

In the assembly control, there basically two operation: Insert Component and Mate (to set the relations of the components)

Reference

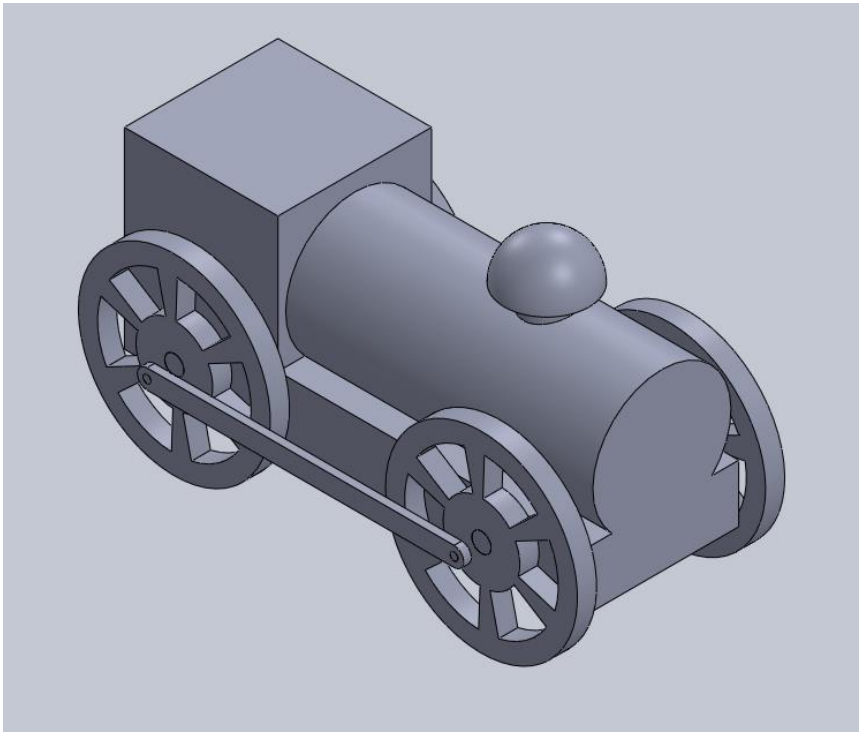
- *INTRODUCING SOLIDWORKS*: P.54 – 66
- *Student's Guide*: P.41 – 56

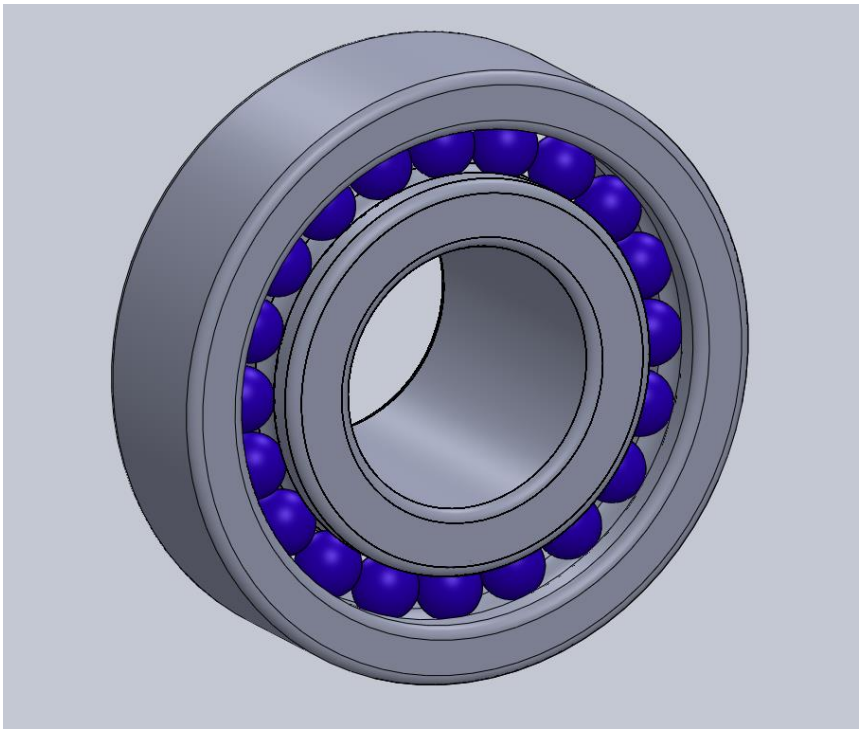
Mate Function

- Standard Mates
  - Coincident
  - Parallel
  - Perpendicular
  - Tangent
  - Concentric
  - Lock
  - Distance
  - Angle
- Advanced Mates
  - Profile Center
  - Symmetric
  - Width
  - Path Mate
  - Linear/Linear Coupler
  - Distance
  - Angle
  - Limit
- Mechanical Mates
  - Cam
  - Slot
  - Hinge
  - Gear
  - Rack Pinion
  - Screw
  - Universal Joint

Exercise

Please assemble the toy train engine and the bearing examples.



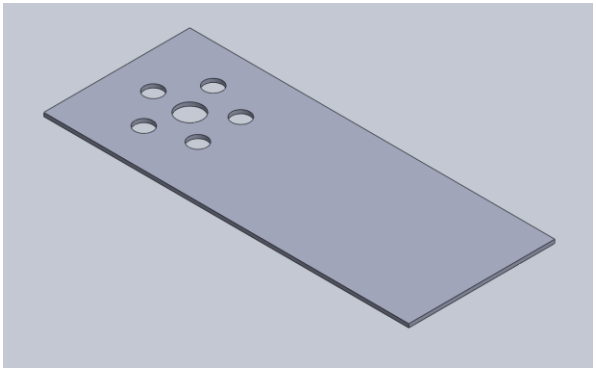


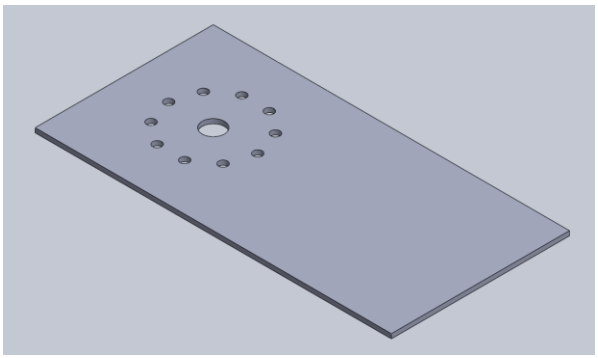
Chapter 3 Design Configuration and Design Table

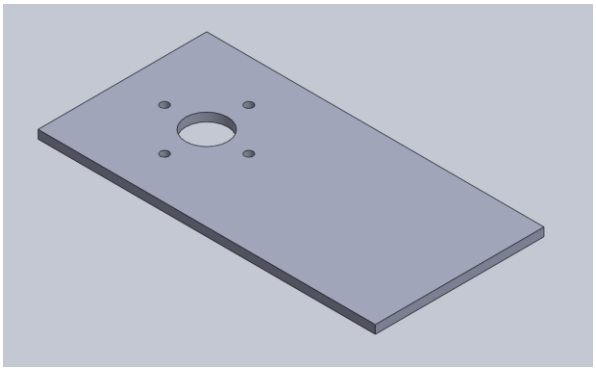
Objective

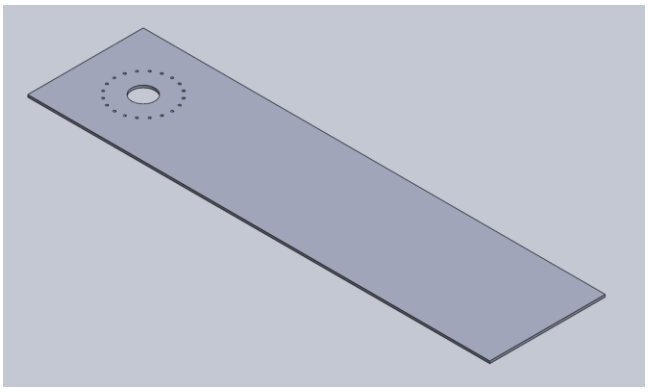
- Create different configurations in SOLIDWORKS
- Create design table in SOLIDWORKS
- Understand the control logic of design table

Example









Exercise

Create a design table to control the configuration of a Lego Part (Please search in the internet about the dimension of a Lego Part).

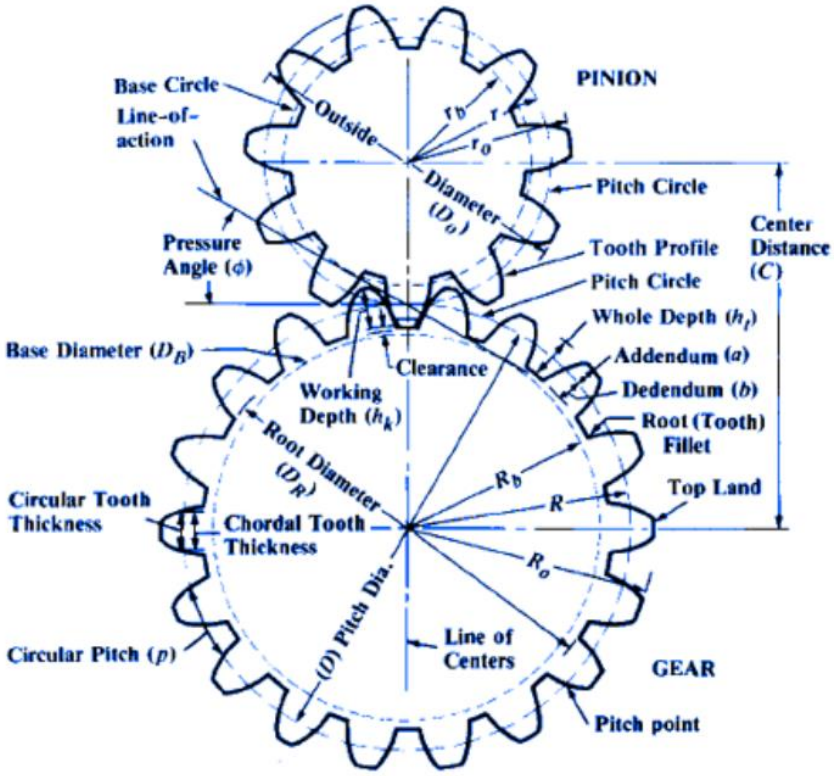
Chapter 4 Gear Design in SOLIDWORKS

Objective

- Create the basic form of gear in SOLIDWORKS
- Build the template of gears to generate different size of gear

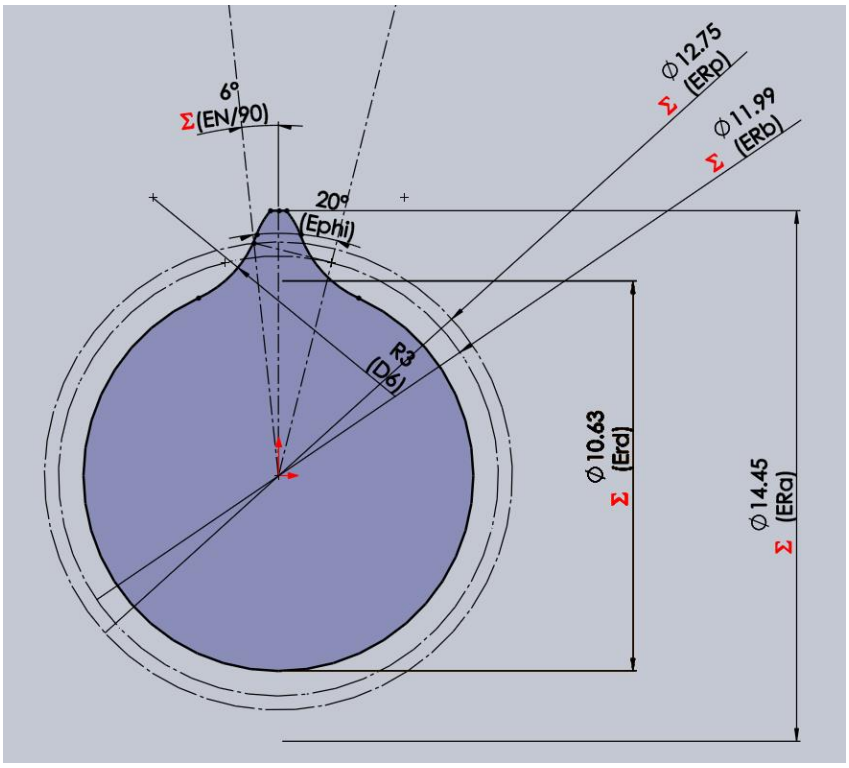
Concept of gear

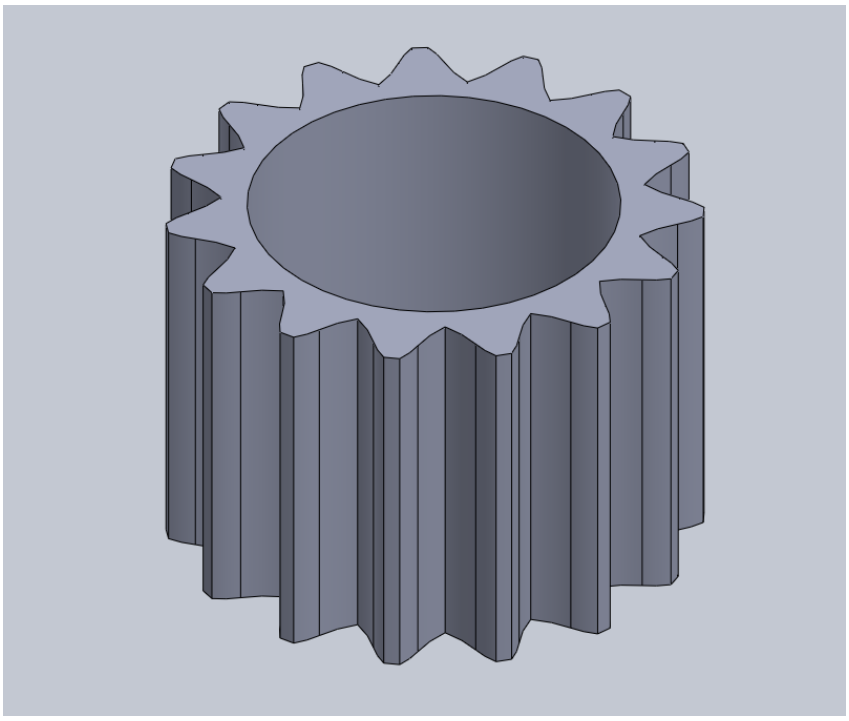
There are a couple ways to define a gear and we are going to cover one of the simplest ways. In the method that we are going to use, we have to define the Module, Number of Teeth and Pressure Angle.



Name	Value / Equation	Evaluates to
Global Variables		
"N"	= 15	15
"phi"	= 20	20
"m"	= 1.7	1.7
"Rp"	= "m" * "N" / 2	12.75
"Rb"	= "Rp" * 0.94	11.985mm
"Ra"	= "Rp" + "m"	14.45
"Rd"	= "Rp" - 1.25 * "m"	10.625

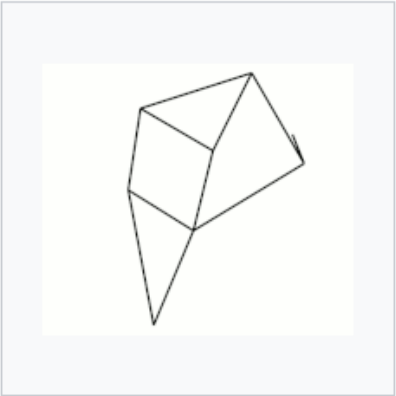
Example

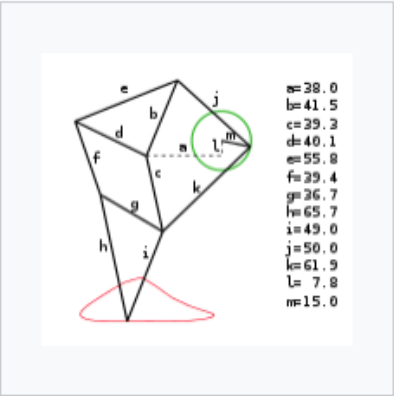


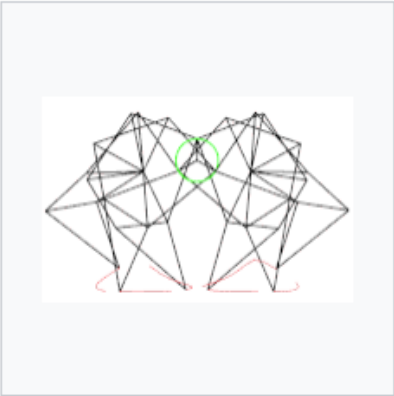


Exercise

Jansen's linkage with gear



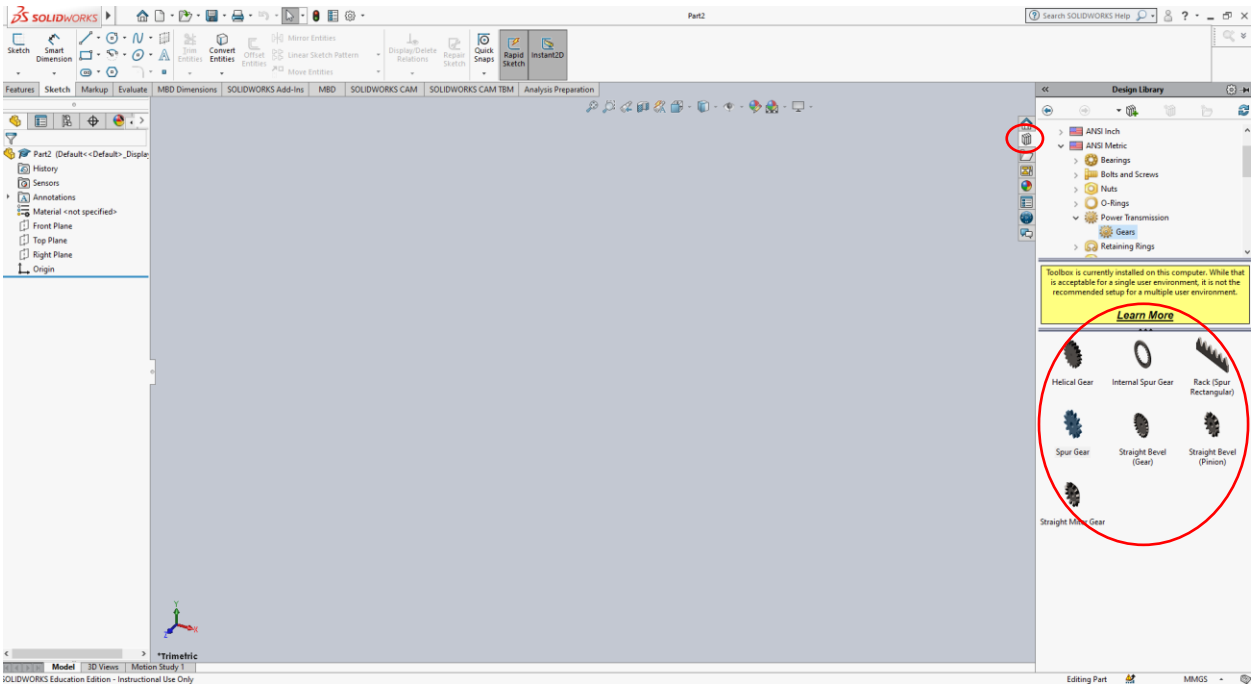




Please create a model for the Jansen's linkage. The physical model will be provided as reference in the class.

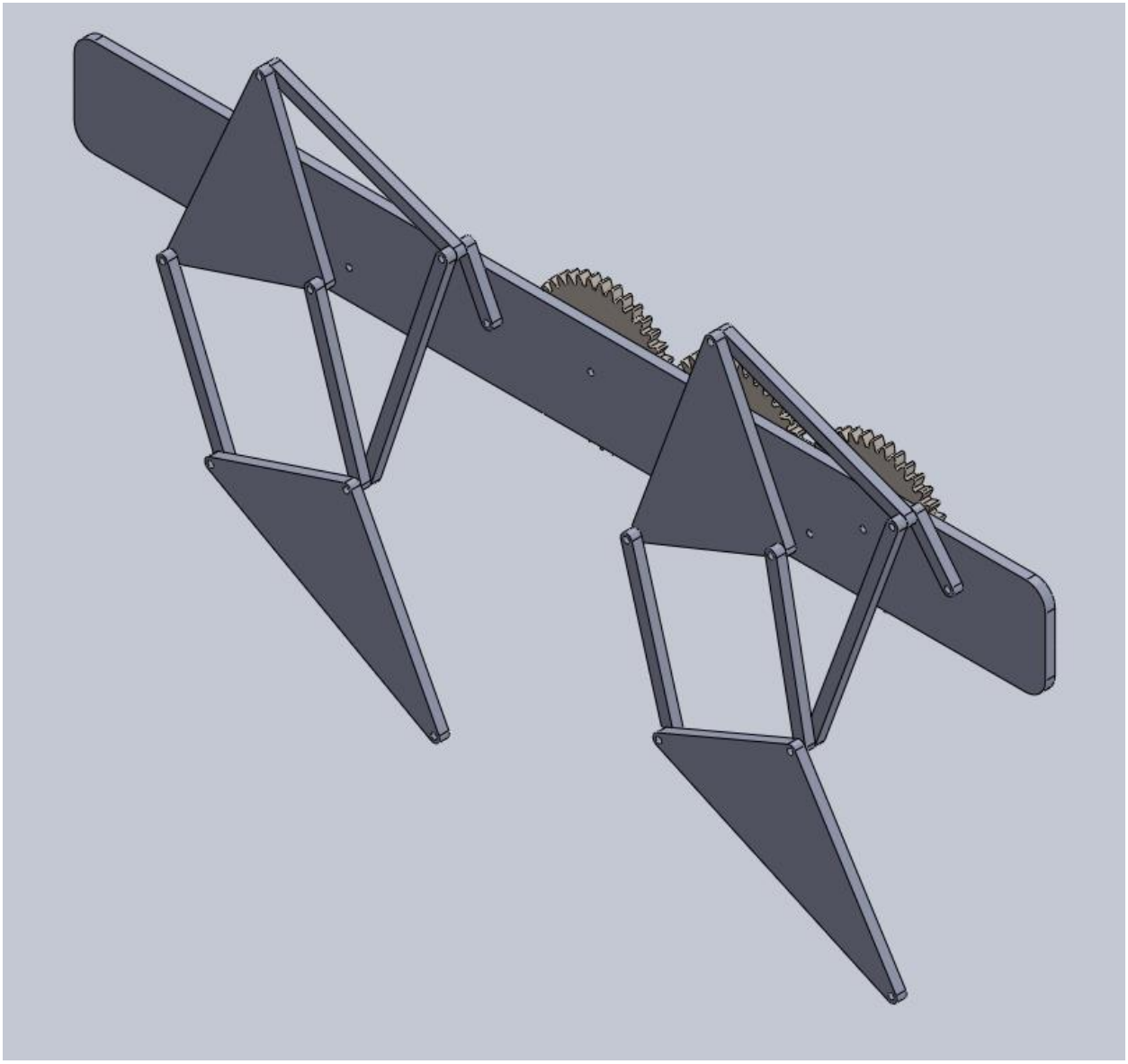
Standard Gears and tool

In fact, standard gear library is available in SOLIWORKS. However, you still have to input the control parameters like Module, Number of Teeth, Pressure Angle, etc. to create the gears.



Exercise

Please create the following model and mate the components.



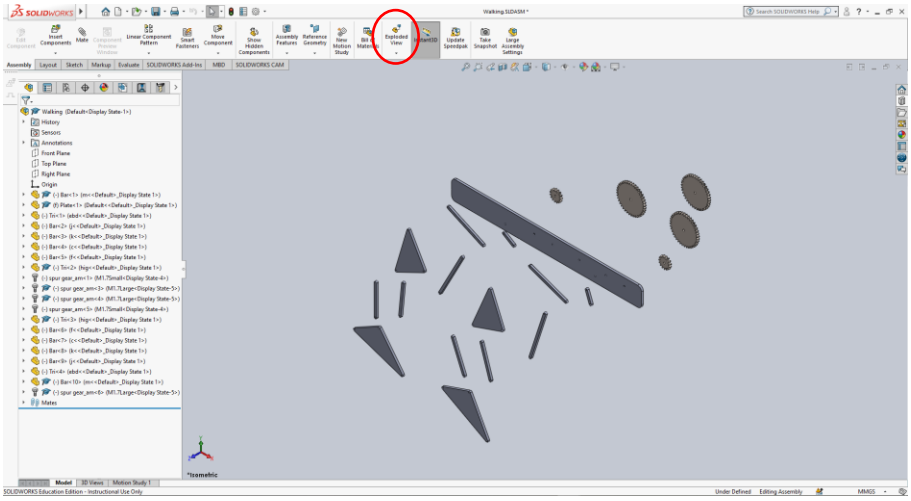
Chapter 5 Exploded View and Simple Animation in SOLIDWORKS

Objective

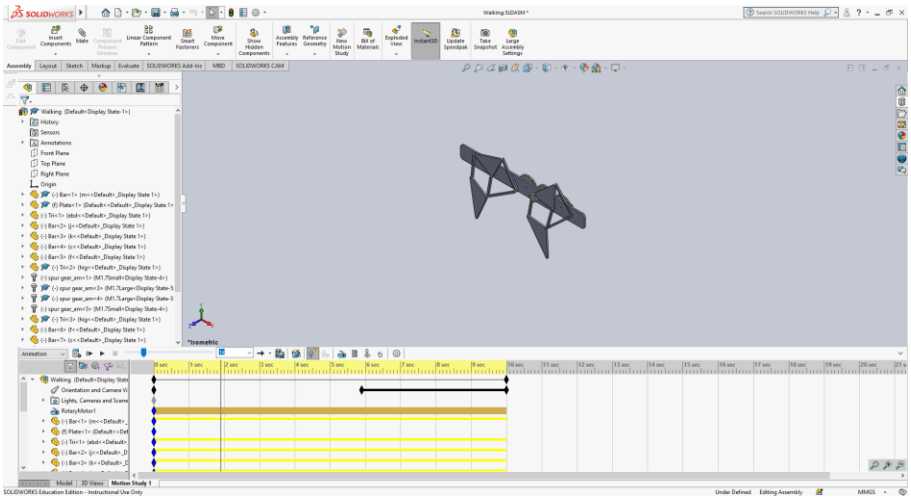
- Create explode view
- Create simple animation

Exploded View

We can simple set the offset or rotate values to define the exploded view. It can be converted to animation easily.



Animation



Exercise

Please create a model to demonstrate the function in this workshop.