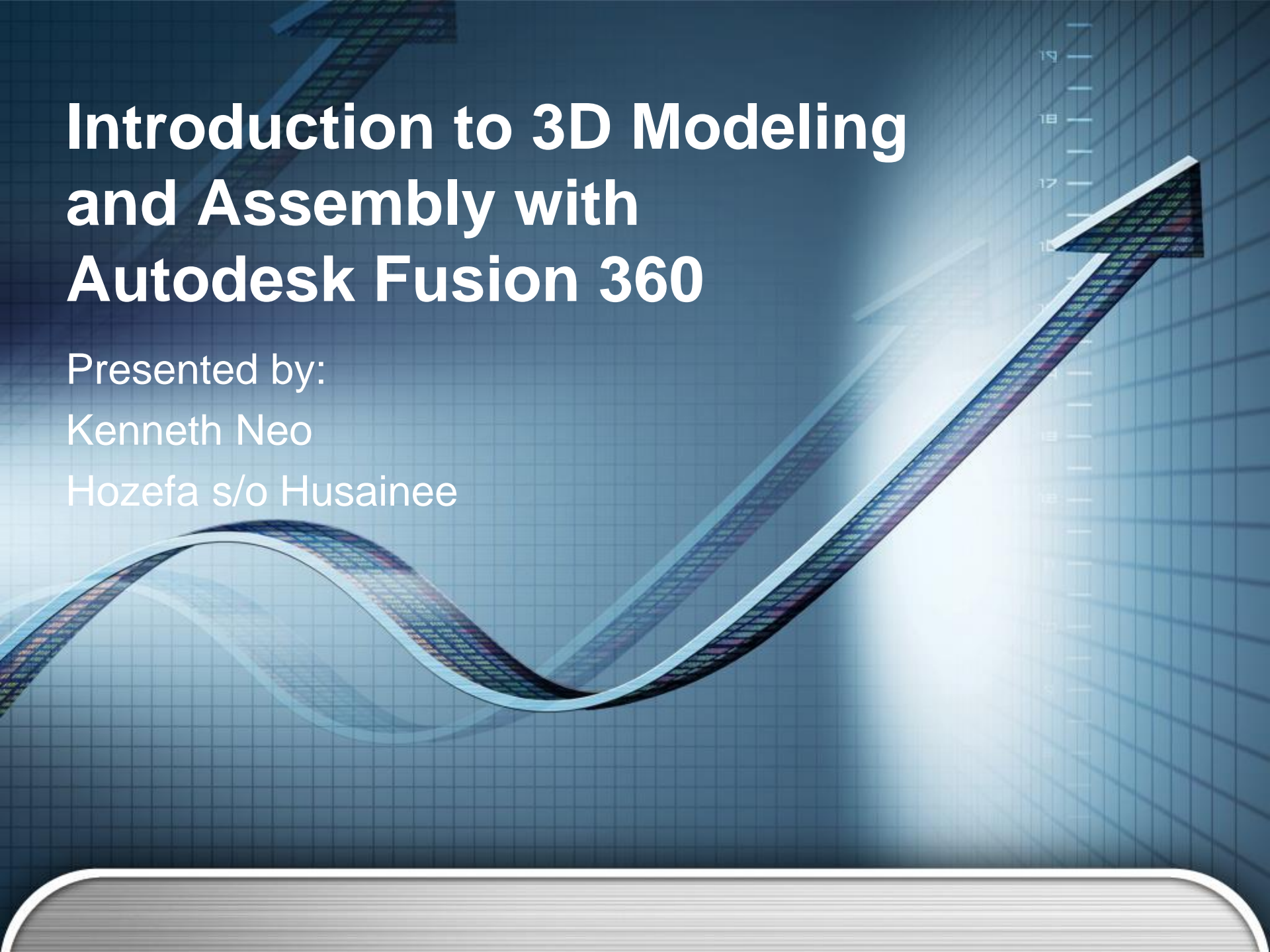


Introduction to 3D Modeling and Assembly with Autodesk Fusion 360

Presented by:

Kenneth Neo

Hozefa s/o Husainee



Lesson Objectives

❖ At the end of this lesson, you will be able to:

- Create 2D parametric sketches using Autodesk Fusion 360
- Build 3D parts using Autodesk Fusion 360
- Create an assembly model
 - Inserting and orientating components in assembly file
 - Applying constrain to components
- Utilize commonly used tools

Contents

- ❖ Introduction
- ❖ 2D Sketch Tools
- ❖ 3D Feature Tools
- ❖ Demonstration
 - Arm
- ❖ Practice

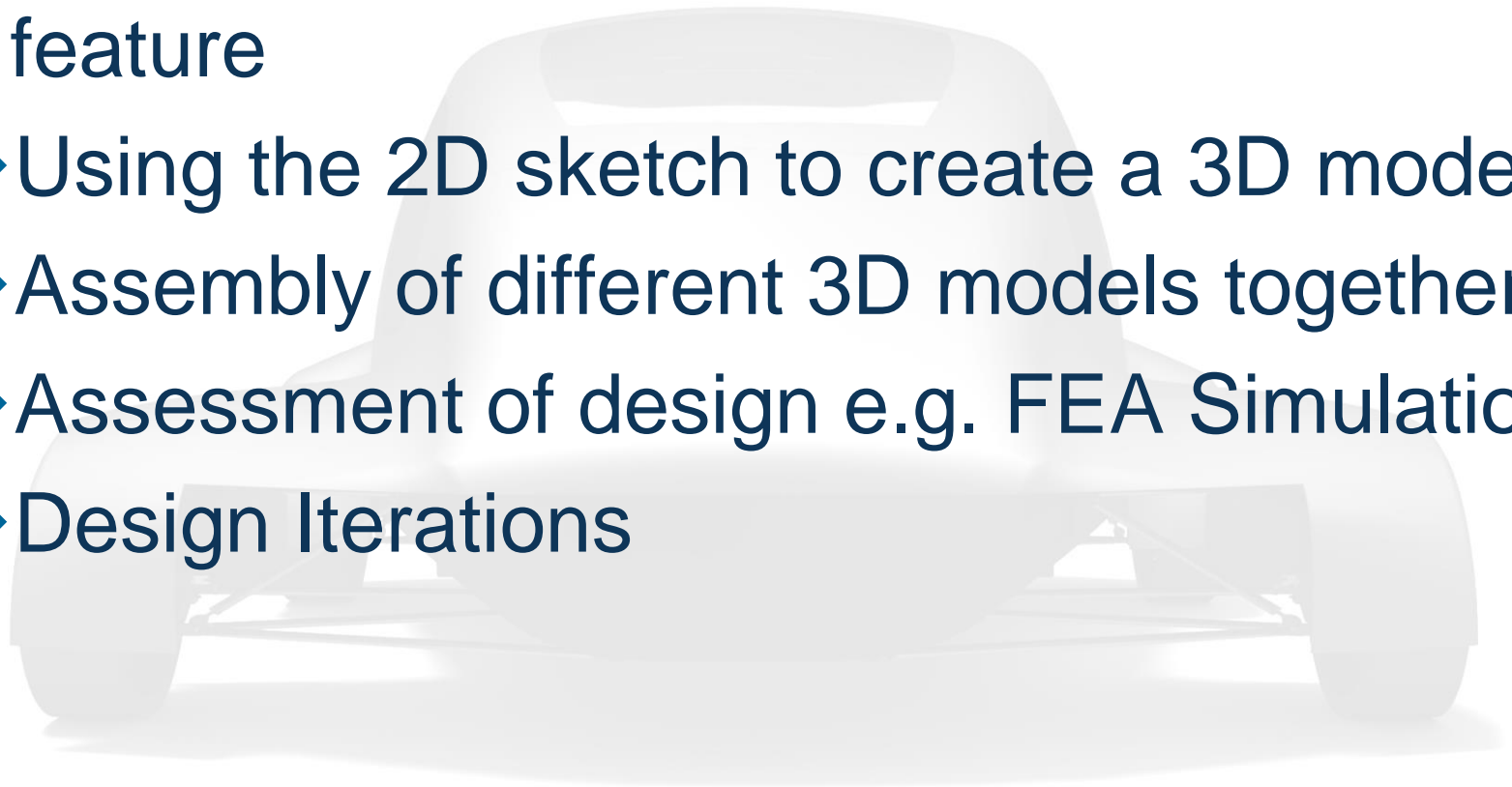
- ❖ Introduction
- ❖ Start new assembly file
- ❖ Place component
- ❖ Positioning
 - Move/ rotate component
 - Apply constrain
- ❖ Demonstration
- ❖ Practice

❖ Computer Aided Design (CAD)

- Using computer software tools to design products
 - Conceptualization of design onto CAD
 - Reduce prototype building
- Ability to translate design to manufacturing operations
- CAD is used extensively in automotive, shipbuilding, aerospace, machine, architectural industries

Part Modeling

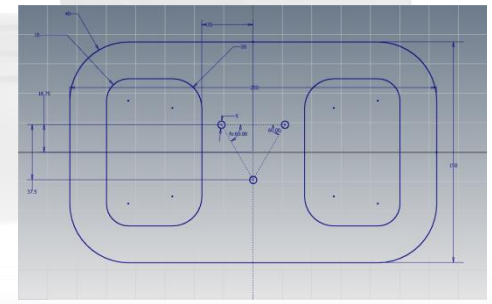
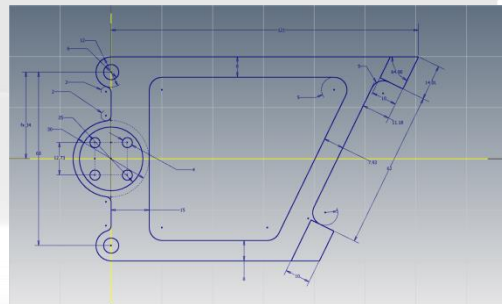
- ❖ Creating 2D parametric sketch as a first feature
- ❖ Using the 2D sketch to create a 3D model
- ❖ Assembly of different 3D models together
- ❖ Assessment of design e.g. FEA Simulation
- ❖ Design Iterations



2D Sketch

❖ 2D parametric sketch

- Using values and constraints to define the size, location and other properties of the sketch
- Defines the basic shape of the 3D model
- 2D sketch can be performed on planes or 2D faces



2D Sketch

Commonly used sketch features

❖ Draw

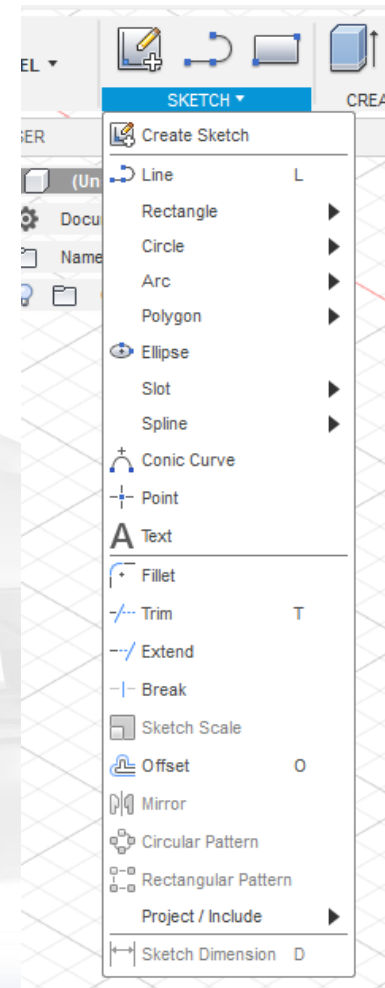
- Line
- Circle
- Arc
- Rectangle
- Spline
- Point

❖ Constrain

- Relation constrain
- Dimension

❖ Pattern

- Circular
- Rectangular
- Mirror



2D Sketch

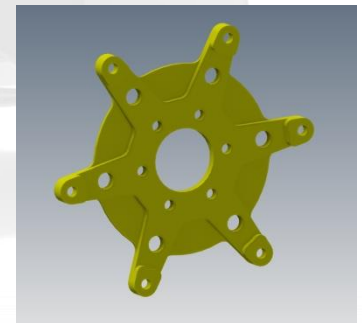
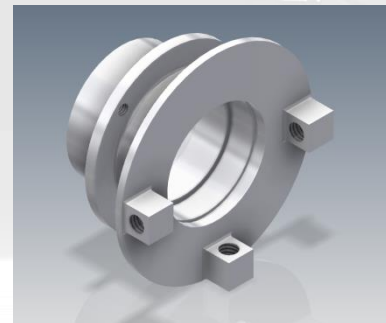
❖ Good Practices

- Fully constrain sketches
 - Prevent accidental dimension change causing assembly or adaptive errors
- Insert most important feature for the components first and constrain them
 - Features where other components are to be mounted on
- Conduct sketch with origin in the center if component have symmetry features
 - Make use of origin planes and axis for rectangular, mirror and circular 3D pattern
 - Use Project Geometry to help you

3D Features

❖ 3D Feature

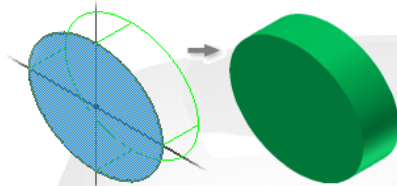
- Built upon closed 2D sketches
- Feature created through “growing” 2D sketch/s along a guided path or around an axis



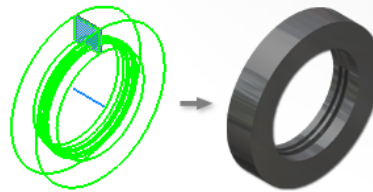
3D Features

❖ Commonly used 3D features

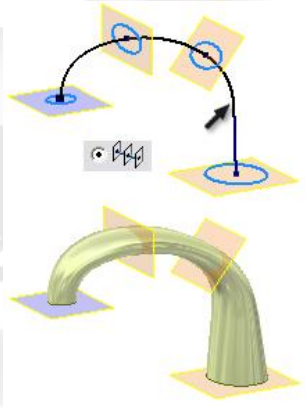
- Extrude



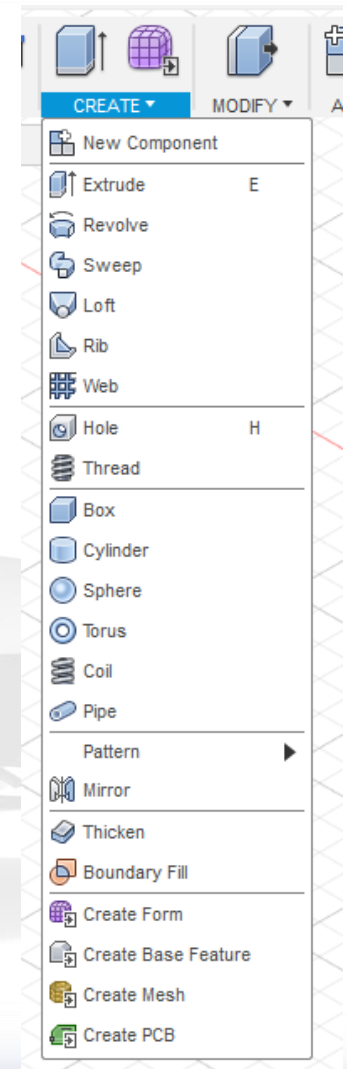
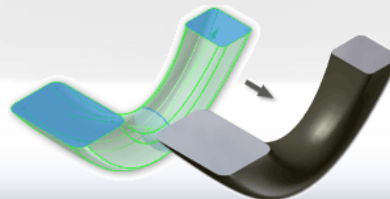
- Revolve



- Loft



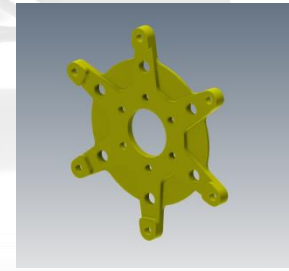
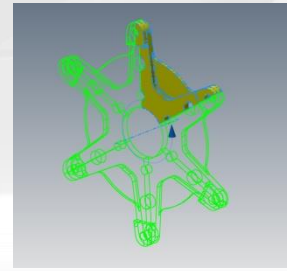
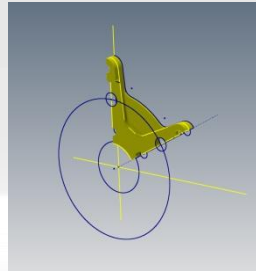
- Sweep



3D Features

❖ Good Practices

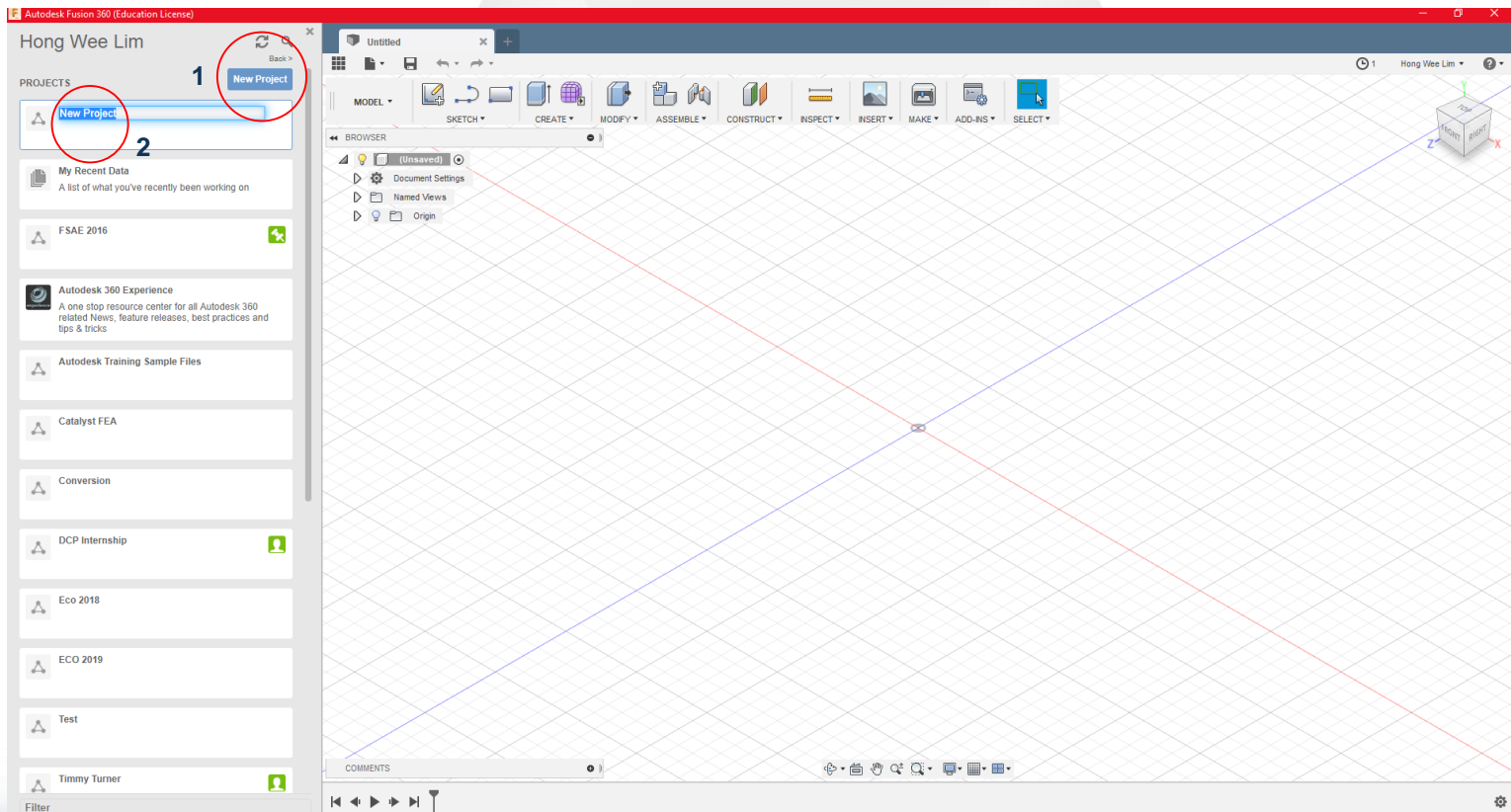
- Create 3D features from closed sketch
 - Open sketch create extruded surfaces
- Create base feature with origin in the center if there are symmetry features
 - Make use of the origin planes and axis for rectangular, mirror and circular 3D pattern
- For axis symmetry components, sketch a portion of the component and make use of circular pattern to create the whole component



2D Sketch

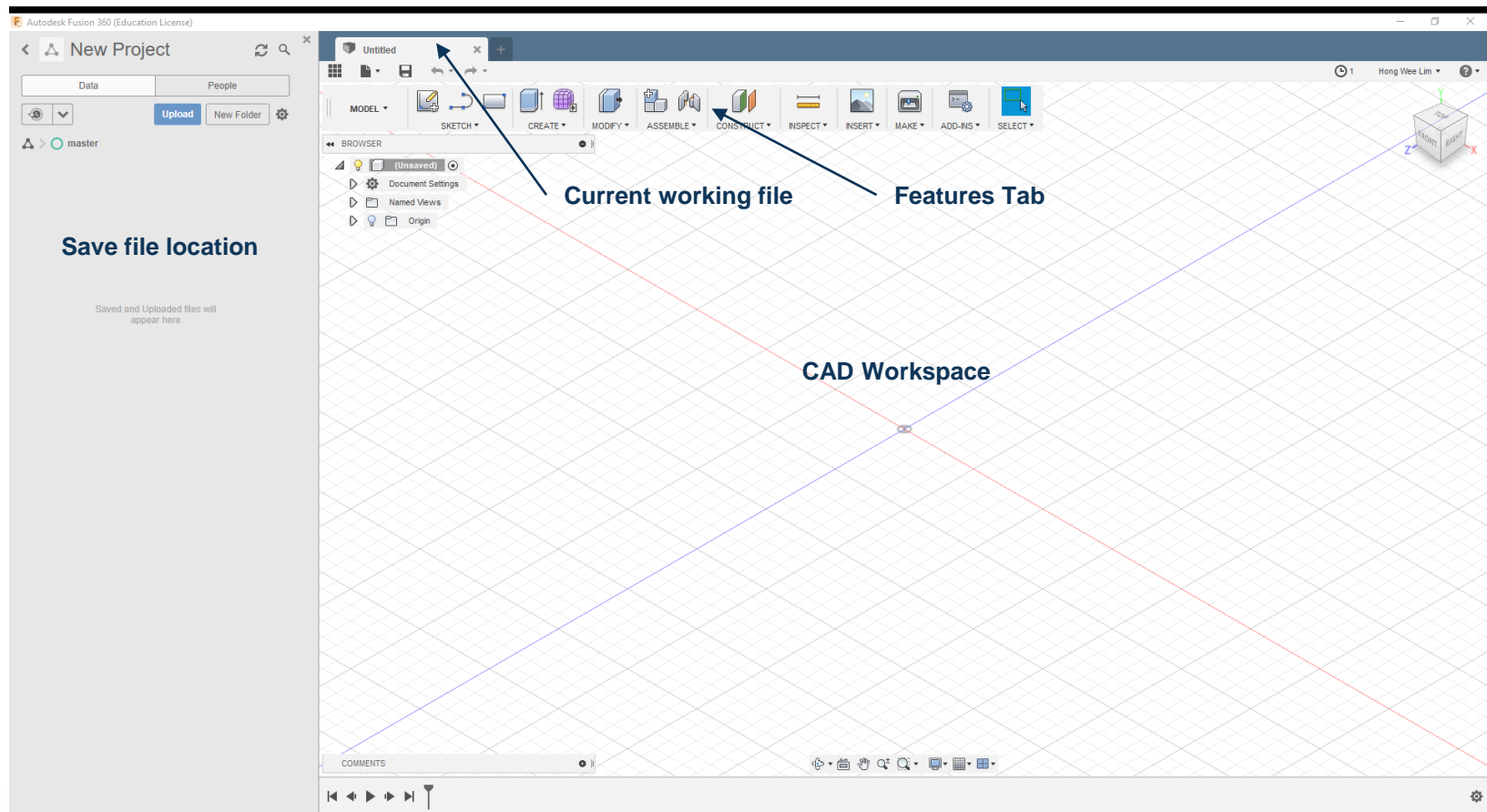
❖ Creating a new project

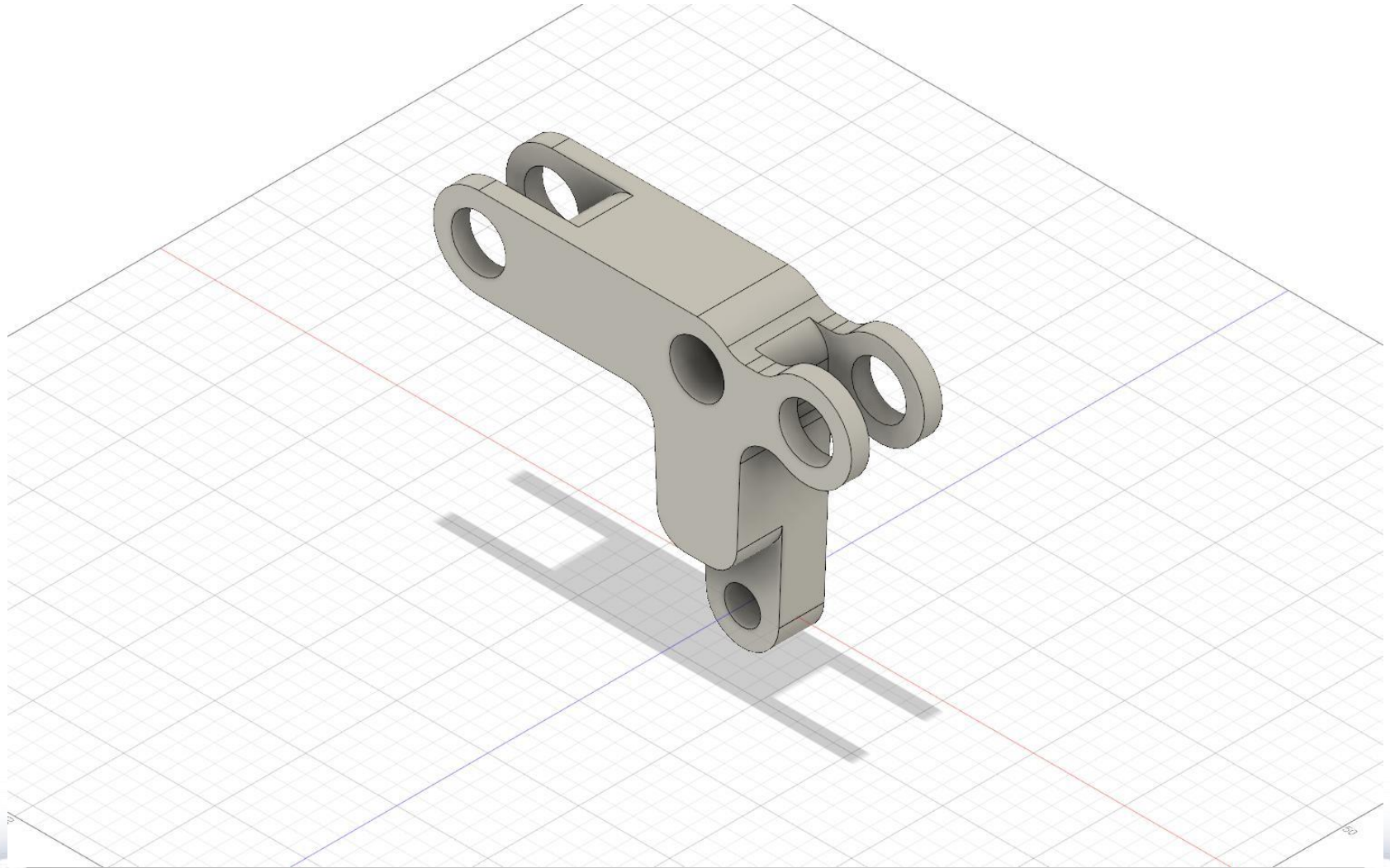
- Launch Fusion 360
- Create new account using your NUS email and login
- Select New Project → Name the Project → Select Named Project



2D Sketch

❖ Fusion 360 layout

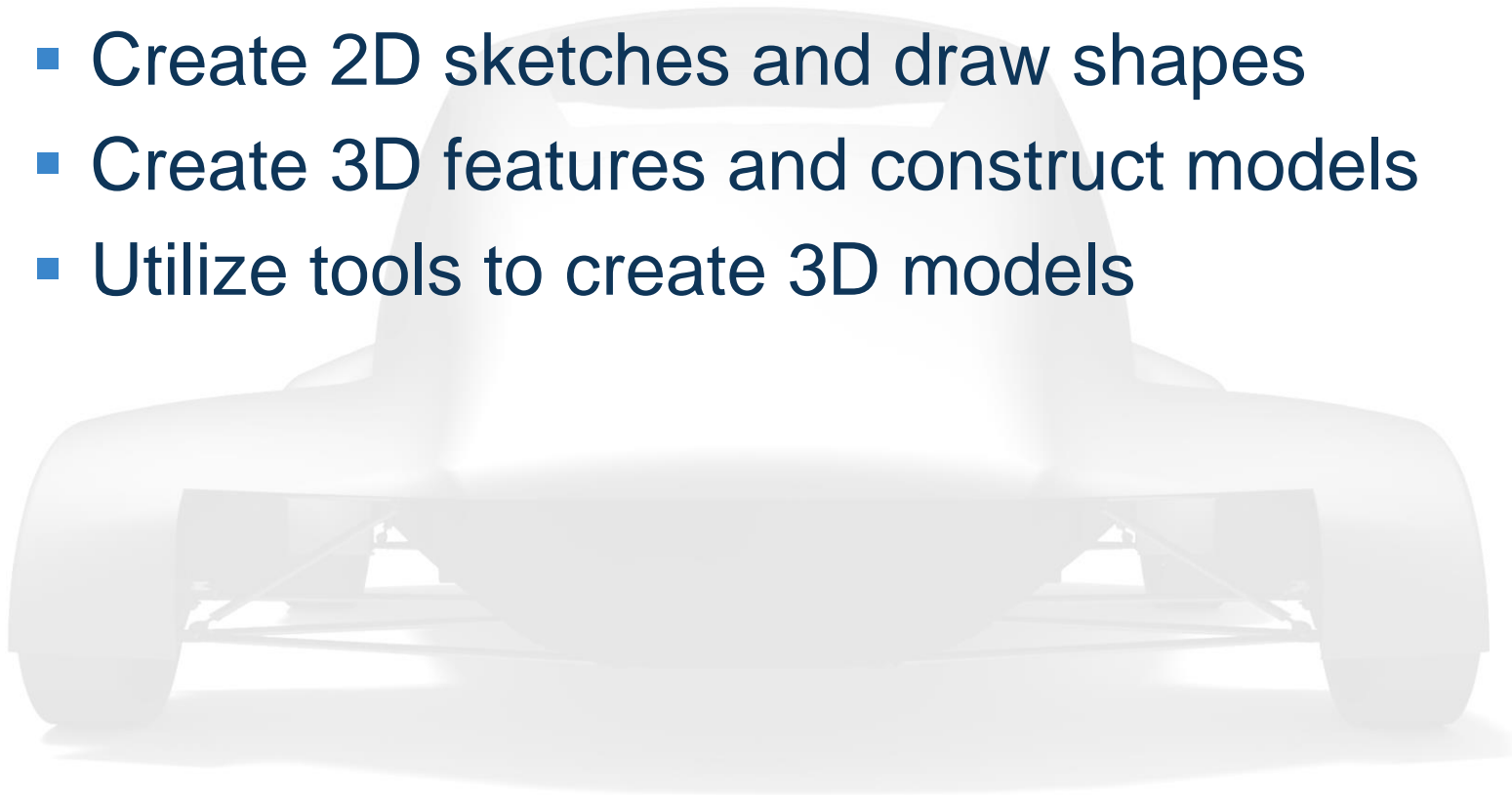




Conclusion

❖ The class is now able to

- Create 2D sketches and draw shapes
- Create 3D features and construct models
- Utilize tools to create 3D models



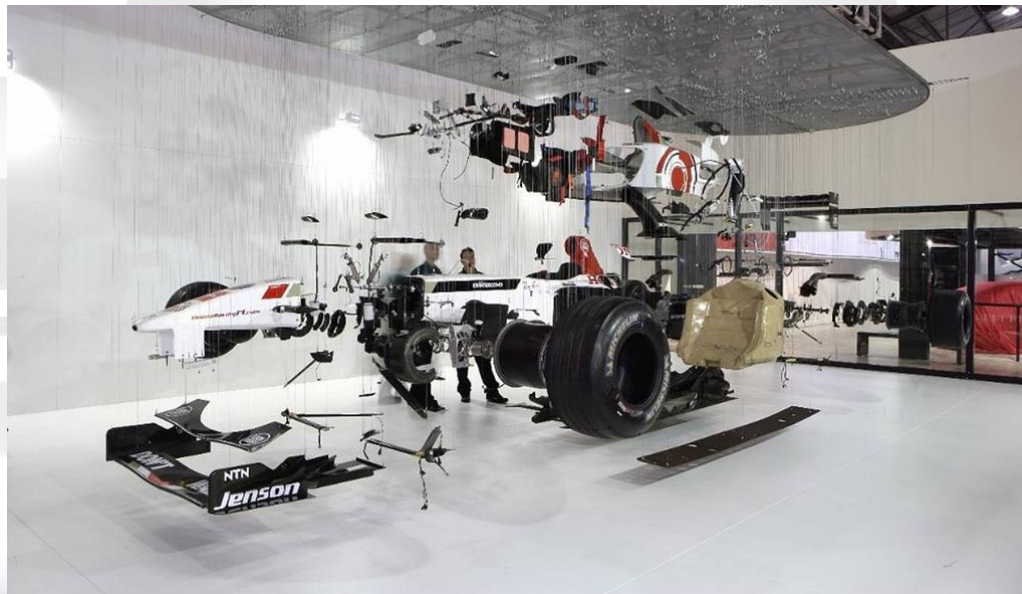
The background features a dark blue grid. A large, stylized arrow points from the bottom left towards the top right. The arrow's shaft is a wavy, multi-colored ribbon (blue, green, yellow, red) that appears to be made of a woven or digital fabric. The arrowhead is a solid dark blue triangle. On the right side, there are vertical white tick marks and numbers: 16, 17, 18, and 19. The text "Thank You" is centered in the upper half, and "Questions?" is centered below it.

Thank You

Questions?

Introduction

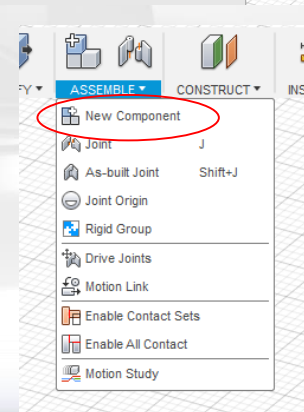
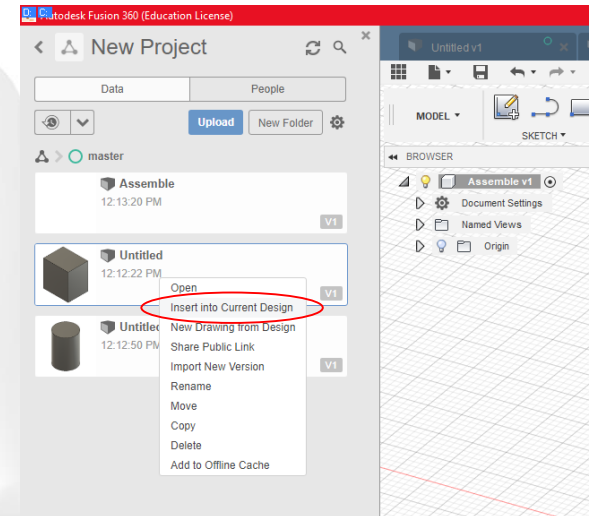
- ❖ A lot of items we use daily are assemblies!
- ❖ Assembly modeling combines parts and sub assemblies to form a new single assembly



Place Components

❖ Inserting Components

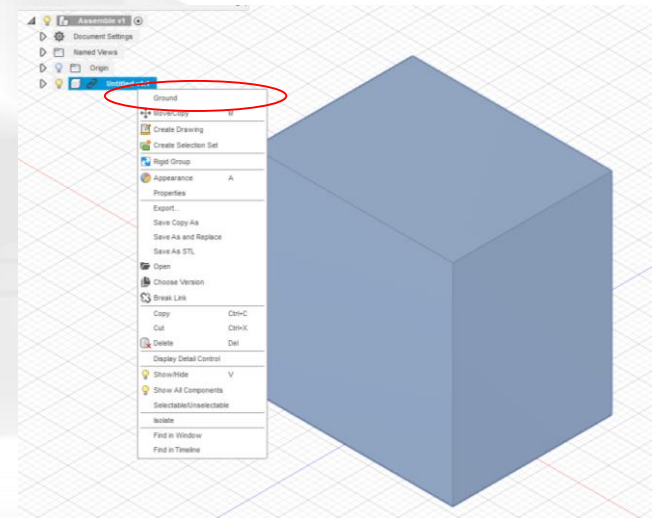
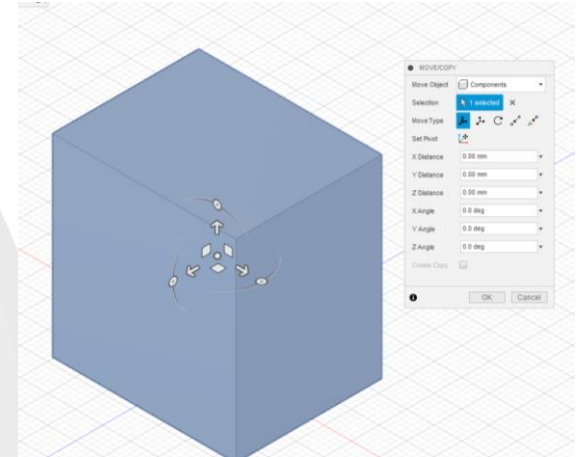
- Inserting component(s) in an assembly file
 - Insert a part or an assembly file
 - Create a new component from the assembly file



Move/ Rotate / Position

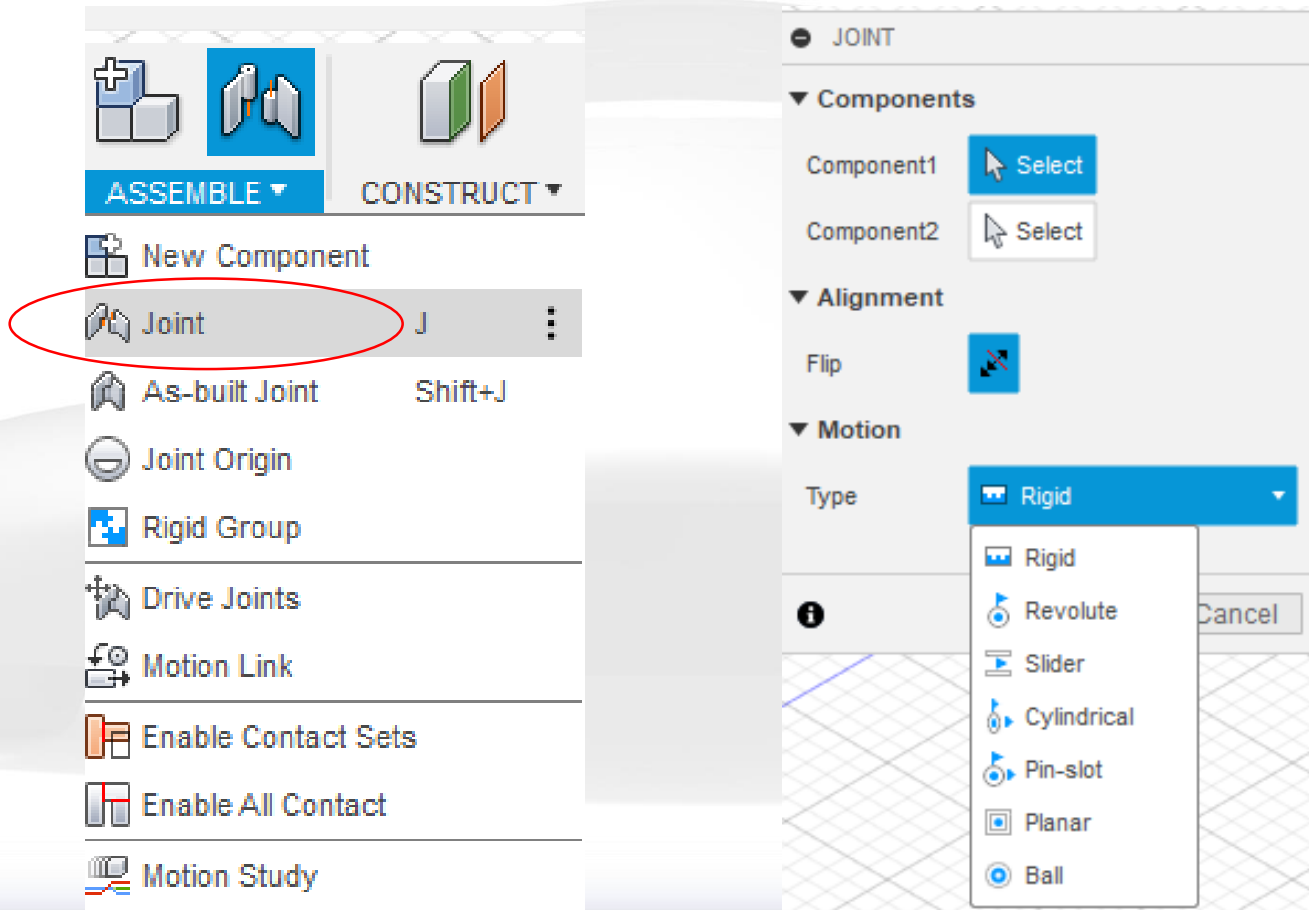
❖ Move/ Copy windows after insertion of component

- Adjust location and orientation
- Ground component (especially first inserted component)
- Prevent floating



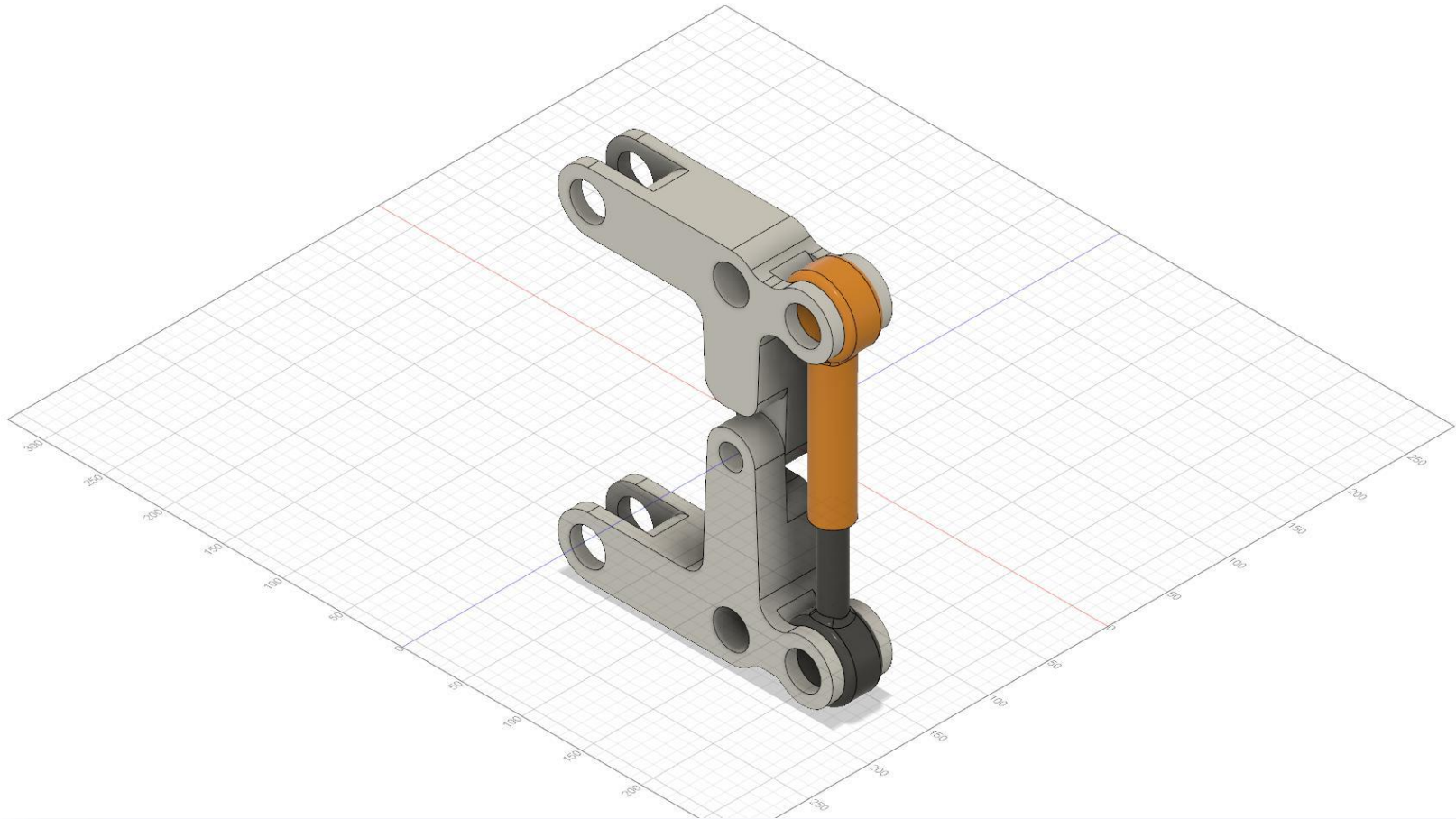
Joint

❖ Allows components to fit together



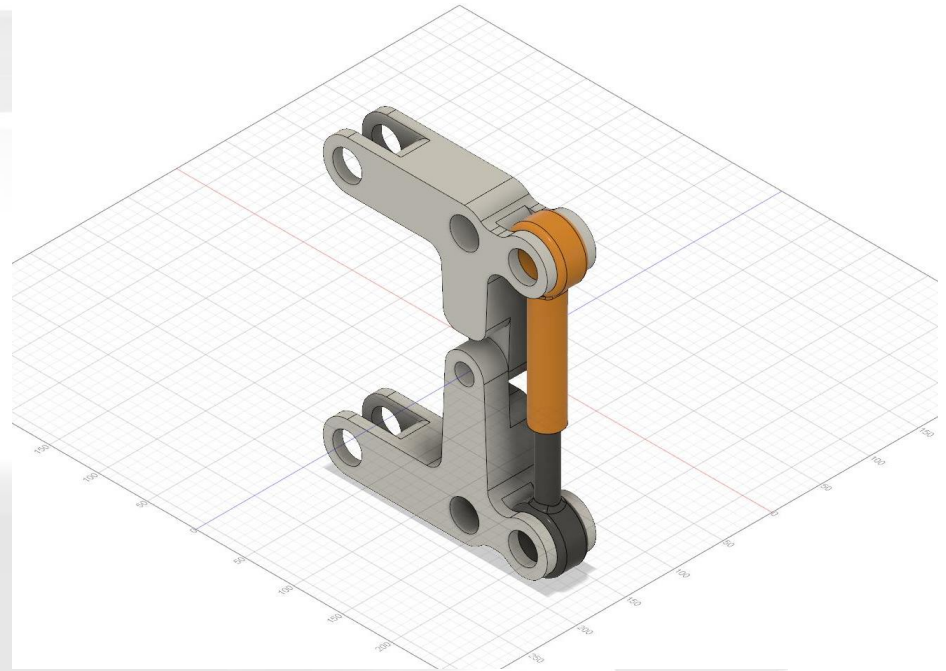
Demonstration

❖ Piston Linkage



Practice

- ❖ Locate your Project Folder.
- ❖ Open “Piston Linkage”.
- ❖ Complete the assembly by inserting the following components
 - Arm
- ❖ Apply constraints such that all components are constraint as shown in the picture
- ❖ Save the assembly.



Assemblies

❖ Good Practices

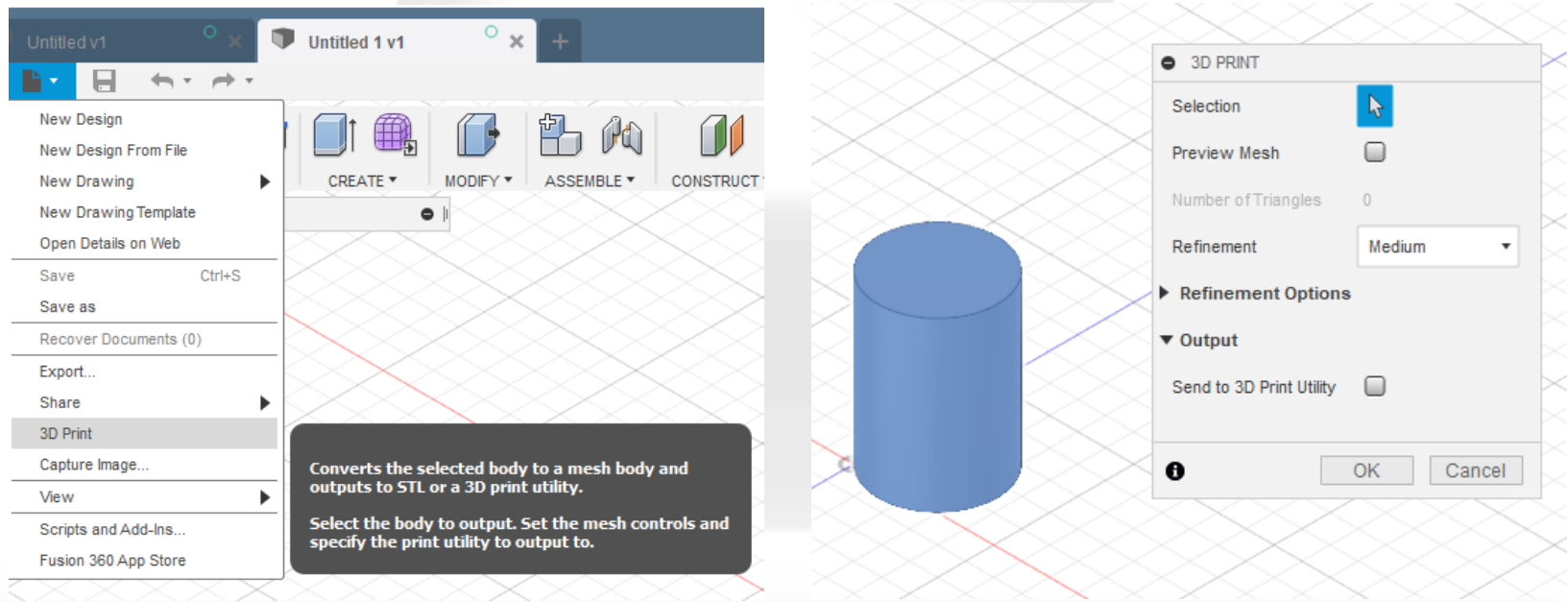
- Use sub assemblies as much as possible
 - Easy to perform localize modification or troubleshoot
 - Minimize rebuilding time after modification



File Conversions

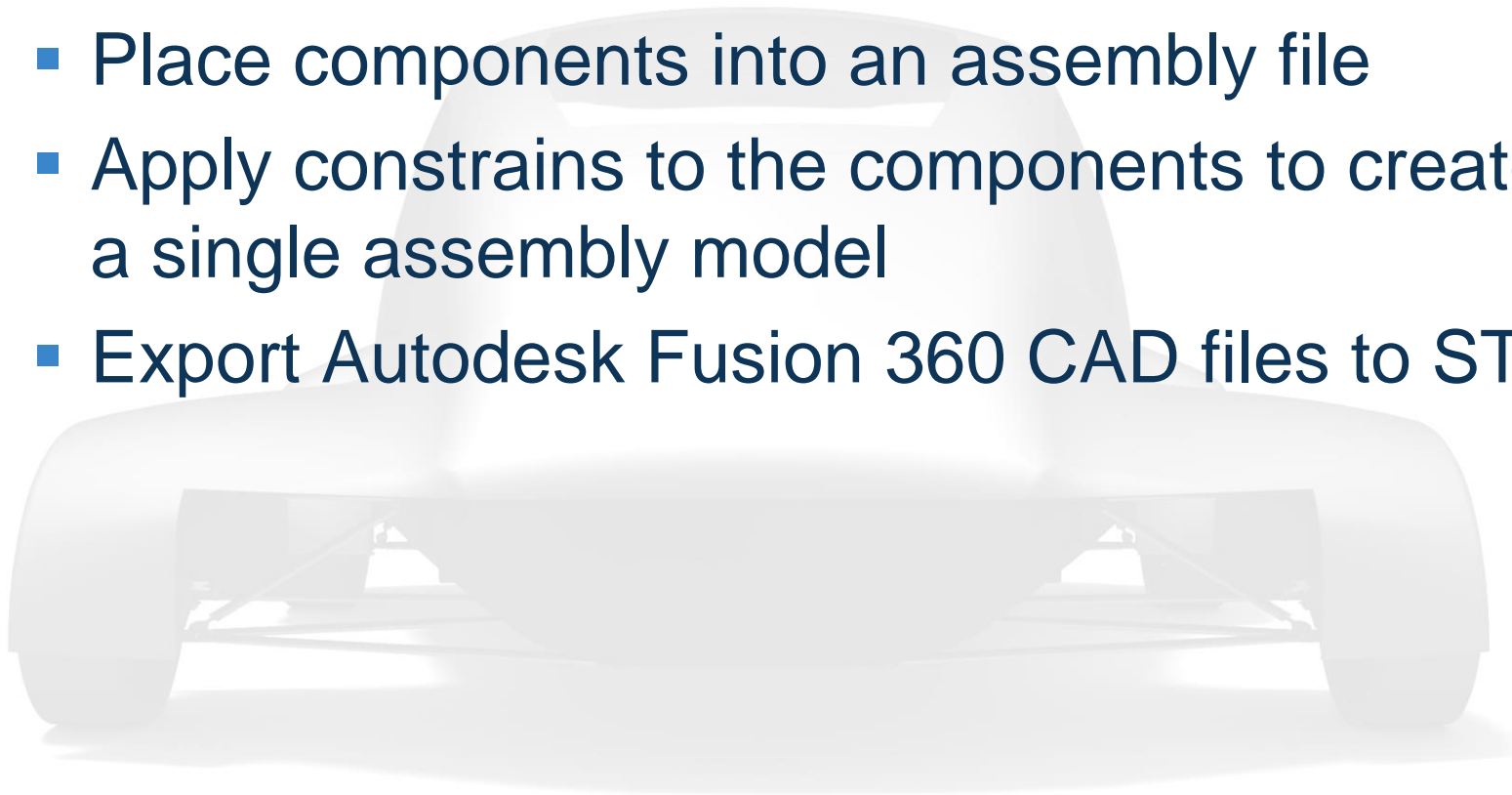
❖ STL format

- Primarily used for 3D printing
- Go to File → 3D Print
- Select component to be printed and click OK



Conclusion

- ❖ The class is now able to
 - Place components into an assembly file
 - Apply constraints to the components to create a single assembly model
 - Export Autodesk Fusion 360 CAD files to STL.



[illegible]