

SolidWorks Essentials

Course details and prerequisites:

Duration: 1 Days
Type: Instructor Led

Less 1: Introduction to Sketching

- Create a new part.
- Insert a new sketch.
- Add sketch geometry.
- Establish sketch relations between pieces of geometry.
- Understand the state of the sketch.
- Use sketch tools to add fillets.
- Extrude the sketch into a solid

Lesson 2: Basic Part Modeling

- Choose the best profile for sketching.
- Choose the proper sketch plane.
- Create a new part.
- Create a sketch.
- Extrude a sketch as a boss.
- Extrude a sketch as a cut.
- Create Hole Wizard holes.
- Insert fillets on a solid.
- Make a basic drawing of a part.
- Make a change to a dimension.
- Demonstrate the associativity between the model and its drawings.

ติดต่อสอบถาม : คุณสง่า คงแก้ว (ไอ้)
ปริญญาตรี : วิศวกรรมเครื่องกล-ลาดกระบัง
ปริญญาโท : สถาบันเทคโนโลยีแห่งเอเชีย (AIT)

Email:sanga_oh@yahoo.com

Line: oh_ait

Tel: 086-7417352

Lesson 3: Modeling a Casting or Forging

- Sketch on a system defined plane or a planar face of a model.
- Use the view display and modification commands.
- Create fully defined sketches through the use of dimensions and geometric relationships.
- Create base and boss features by extrusion.
- Create cut features by extrusion.
- Copy and paste features.
- Create constant radius fillets.
- Edit the definition and parameter of a feature and regenerate the model.
- Use Up To Next and Mid Plane end conditions to capture design intent.
- Use symmetry in the sketch.

Lesson 4 : Patterning

- Create a linear pattern
- Add a circular pattern

Lesson 5 : Revolved Features

- Create a mirror pattern
- Create revolved features.
- Apply special dimensioning techniques to sketches for revolved features.
- Create a sweep feature.
- Calculate the physical properties of a part.
- Perform rudimentary, first pass stress analysis

Lesson 5: Loft feature

- Creating planes
- Sketching, copying, and pasting the profiles
- Creating a solid by connecting the profiles (lofting)

Lesson 6: Shelling

- Create a thin wall

Lesson 7: Configurations of Parts

- Use configurations to represent different versions of a part within a single SolidWorks file.
- Use modify configurations to create and edit configurations.
- Suppress and unsuppressed features.
- Change dimension values by configuration.
- Suppress features by configuration.
- Understand the ramifications of making changes to parts that have configuration.

Lesson 8: Design Tables and Equations

- Link dimension values together to capture design intent.
- Create Equations.
- Automatically create design tables.
- Use existing design tables to create families of parts
- Make detail drawings using more advanced types of drawing views.

Lesson 9: Bottom-Up Assembly Modeling

- Create a new assembly.
- Insert components into an assembly using all available techniques.
- Add mating relationships between components.
- Utilize the assembly-specific aspects of the Feature Manager Design tree to manipulate and manage the assembly.
- Insert sub-assemblies
- Use part configurations in an assembly