



KG REDDY

College of Engineering
& Technology

Certificate Course in Mechanical Engineering with Specialization “SOLID WORKS”

Held On

05th March to 09th March 2018



Department of Mechanical Engineering,
KG Reddy College of Engineering & Technology

Chilkur (Village), Moinabad (Mandal), Hyderabad RR Dist-501504

Course coordinator

Principal

Principal
KG Reddy College of Engineering & Technology
Chilkur (V), Moinabad (M).
R.R. Dist., Telangana.

SUMMARY REPORT ON SOLID WORKS

About the Course

The certificate course on Solid works is concluded its work successfully by department of Mechanical Engineering (ME) in KG Reddy college of Engineering and technology (KGR CET), Hyderabad, Telangana. This course is a forum to bring together students to discuss innovative ideas and diverse topics of this course on next generation of information technologies. Department has taken a new step for students to improve the quality of study through this course and become most wide scale , extensive, spectacular event in ME. The course was held in two locations of the department (a) Department E-learning room and (b) Department class.

Solid Works is a solid modeling computer-aided design (CAD) and computer-aided engineering (CAE) computer program that runs primarily on Microsoft Windows. While it is possible to run SolidWorks on MacOS, that is not supported by SolidWorks.^[2] SolidWorks is published by Dassault Systèmes. SolidWorks is a solid modeler, and utilizes a parametric feature-based approach which was initially developed by PTC (Creo/Pro-Engineer) to create models and assemblies. The software is written on Parasolid-kernel.

This course is absolutely practical oriented course which is helped to student for making their carrier through database in any industry. The students of 2nd year 2nd semester have been benefited in many ways from this course. More than 80 students have joined in this course as their own interest and completed this course. The trainer taught to students very nice with real time example and sharing his knowledge to develop technical skill in industry.

Objectives of the course

Solid Works is a 3D solid modeling package which allows users to develop full solid models in a simulated environment for both design and analysis. In Solid Works, it sketch ideas and experiment with different designs to create 3D models. Solid Works is used by students, Designers, Engineers, and other professionals to produce simple and complex parts, assemblies, and drawings. Designing in a modeling package such as Solid Works is beneficial because it saves time, effort, and money that would otherwise be spent prototyping the design.

Scope of the Course

- The scope of SolidWorks/CAD/CAM professionals in India are plethora. As India is considered as the software industry hub in the whole world.
- Passion for this profession is the must and most valuable skill required.
- Day by day, the demands of this profession are booming at a rapid pace.
- The term CAD CAM implies that an engineer can use the system both for designing a product and for controlling manufacturing process.
- CAD CAM is extensively used in the fields of engineering, architecture and entertainment medias.
- Industrial designers now use computer graphics to do work that was previously done with pencil and paper.

- Lucrative opportunities exist with original equipment manufacturers in the automobile and consumer product sector, in their design, development and manufacturing divisions.
- The work could relate to tool rooms, Mould design, CNC machines and CNC machine shops.
- Today the use of CAD CAM CNC tools has become a necessity in organisations over industry verticals such as automotive, aerospace and utilities.

Benefits

1. **Concurrent engineering (CE)**– Engineering and manufacturing process are enabled simultaneously from shared Solidworks data.
2. **Higher quality** – Due to increased efficiency resulting from the ability to explore a greater number of design iterations during product development.
3. **Lower unit costs** – Due to reduced development and prototype expenses.
4. **Rapid prototyping (RP)** – Solidworks models can be used to produce prototypes from Stereolithography and other RP technologies.
5. **Personnel development** – Solidworks technology provides a challenging environment for employees.
6. **Personnel advancement** – A variety of positions regarding the management and supervision of Solidworks become available to advance employee careers.
7. **Identify and eliminate inefficiencies** – Solidworks develops opportunities for the elimination of inherent inefficiencies in existing work flows and/or practices.
8. **Increased workload capacity** – Efficient use of Solidworks allows the production of more work while maintaining current staff levels.
9. **Greater feedback and control of production operations** – Solidworks enables NC tool paths to be generated, updated, and verified automatically with little human intervention.
10. **Improved overall communications** – Solidworks enable a shift from the traditional paper based design and manufacturing system to a electronic paperless one.
11. **Increased accuracy of MRP data** – Solidworks data files can be easily linked and managed by MRP software.
12. **Increased design flexibility** – Solidworks offers a more robust set of tools and methods to modify designs.
13. **Increased design data integrity** – With a single Solidworks model supporting all downstream processes, changes are reflected quickly and accurately.

Summary of Participants

- (a) Number of students attended this course: 53
(b) Number of student attend the exam: 53
(c) Number of certificate issued: 53

Day-1
05/02/18

Time: 09:00 AM to 10:00 AM

Inauguration of certificate course

The first day of certificate course started with welcoming and opening ceremony at the KGR CET conference Hall. The following dignitaries were representatives of the certificate course who were addressed and pointed out the importance on course with short welcoming speeches.

Welcome addressed by Dr. P. Pravuraj, HOD, H&S, KGR CET
About the certificate course by Principal Dr. R. S. Jahagirdar, KGR CET.
Importance of this course by expert trainer Mr. G. Ganesh, Hyderabad
Interaction with 2nd year 2nd semester students

Time: 10:00 AM to 04:15 PM

Solid Works is a solid modeling computer-aided design (CAD) and computer-aided engineering (CAE) computer program that runs primarily on Microsoft Windows. While it is possible to run SolidWorks on MacOS, that is not supported by SolidWorks.^[2] SolidWorks is published by Dassault Systèmes.

SolidWorks is a solid modeler, and utilizes a parametric feature-based approach which was initially developed by PTC (Creo/Pro-Engineer) to create models and assemblies. The software is written on Parasolid-kernel.

Parameters refer to constraints whose values determine the shape or geometry of the model or assembly. Parameters can be either numeric parameters, such as line lengths or circle diameters, or geometric parameters, such as tangent, parallel, concentric, horizontal or vertical, etc. Numeric parameters can be associated with each other through the use of relations, which allows them to capture design intent.

Design intent is how the creator of the part wants it to respond to changes and updates. For example, you would want the hole at the top of a beverage can to stay at the top surface, regardless of the height or size of the can. SolidWorks allows the user to specify that the hole is a feature on the top surface, and will then honor their design intent no matter what height they later assign to the can.

Features refer to the building blocks of the part. They are the shapes and operations that construct the part. Shape-based features typically begin with a 2D or 3D sketch of shapes such as bosses, holes, slots, etc. This shape is then extruded or cut to add or remove material from the part. Operation-based features are not sketch-based, and include features such as fillets, chamfers, shells, applying draft to the faces of a part, etc.

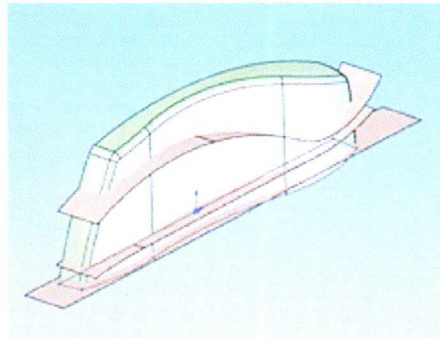


Fig: captured from a SolidWorks top-down design approach

Building a model in SolidWorks usually starts with a 2D sketch (although 3D sketches are available for power users). The sketch consists of geometry such as points, lines, arcs, conics (except the hyperbola), and splines. Dimensions are added to the sketch to define the size and location of the geometry. Relations are used to define attributes such as tangency, parallelism, perpendicularity, and concentricity. The parametric nature of SolidWorks means that the dimensions and relations drive the geometry, not the other way around. The dimensions in the sketch can be controlled independently, or by relationships to other parameters inside or outside of the sketch.

Day-2
06/02/18

In an assembly, the analog to sketch relations are mates. Just as sketch relations define conditions such as tangency, parallelism, and concentricity with respect to sketch geometry, *assembly mates* define equivalent relations with respect to the individual parts or components, allowing the easy construction of assemblies. SolidWorks also includes additional advanced mating features such as gear and cam follower mates, which allow modeled gear assemblies to accurately reproduce the rotational movement of an actual gear train.

Finally, drawings can be created either from parts or assemblies. Views are automatically generated from the solid model, and notes, dimensions and tolerances can then be easily added to the drawing as needed. The drawing module includes most paper sizes and standards (ANSI, ISO, DIN, GOST, JIS, BSI and SAC).

SolidWorks files (previous to version 2015) use the Microsoft Structured Storage file format. This means that there are various files embedded within each SLDDRW (drawing files), SLDPRT (part files), SLDASM (assembly files) file, including preview bitmaps and metadata sub-files. Various third-party tools (see COM Structured Storage) can be used to extract these sub-files, although the subfiles in many cases use proprietary binary file formats.

Day-2
06/02/18

Basic Concepts

Measures distance, angle, and radius in sketches, 3D models, assemblies, or drawings. Also measures the size of and between lines, points, surfaces, and planes.

Sensors

Sensors monitor selected properties of parts and assemblies and alert you when values deviate from the limits you specify.

Equations

Define dimensions using global variables and mathematical functions, and create mathematical relationships between two or more dimensions in parts and assemblies.

Mirror and Opposite-Hand Disambiguation

Various commands are available for mirroring and creating opposite-hand versions of items such as sketch entities, features, parts, and assembly components. The command you need depends on the type of item for which you want to create a mirror or an opposite-hand version.

Rebuild Tools

After you make changes in a model, you rebuild the model to update features and check for errors. Various tools are available to rebuild parts, assemblies, drawings, and sketches.

Industry-Specific Design Tools

The SOLIDWORKS Xperts are tools that help novices use SOLIDWORKS like experts, without having to understand the details of the software. The Xperts take care of the details of software functionality. Using Xperts, you can focus on your design intent while the Xperts focus on what the software should do to achieve it.

Add-Ins

You can load compatible applications into your SOLIDWORKS software, for the current session or at the next startup, or both. The add-in applications must be installed on your computer.

Day-3
07/02/18

SOLIDWORKS Fast Start

To launch more quickly, SOLIDWORKS begins loading components in the background when you start your computer.

Object Linking and Embedding

You can embed an OLE object from another program into the active SOLIDWORKS document.

Recording and Playing Macros

Macros are scripts that let you run operations in the SOLIDWORKS software automatically.

Future Version Components in Earlier Releases

You can open SOLIDWORKS parts and assemblies using Service Pack 5 of the previous release.

SOLIDWORKS API

The SOLIDWORKS Application Programming Interface (API) is a COM programming interface to the SOLIDWORKS software. Functions in the API provide programmers with direct access to SOLIDWORKS functionality.

SOLIDWORKS Task Scheduler

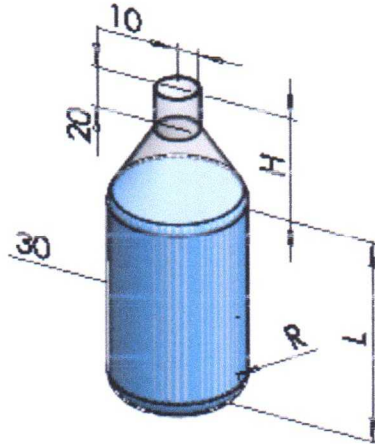
The SOLIDWORKS Task Scheduler lets you set up tasks to perform at a future time.

About SOLIDWORKS

There are two main modes for running a Design Study:

Evaluation It specifies discrete values for each variable and use sensors as constraints. The software runs the study using various combinations of the values and reports the output for each combination.

For example, for this multibody model of a water bottle, you specify values of 75mm, 100mm, and 150mm for the length (**L**); 30mm, 55mm, and 80mm for the height (**H**); and 10mm and 20mm for the radius (**R**). You specify a Volume sensor to monitor the volume of the water body. The Design Study results report the volume of the water for each combination of **L**, **R**, and **H**.




Optimization You specify values for each variable, either as discrete values or as a range. You use sensors as constraints and as goals. The software runs iterations of the values and reports the optimum combination of values to meet your specified goal.

For example, for the model above, you specify a range of 75mm to 150mm for the length (**L**); discrete values 30mm, 55mm, and 80mm for the height (**H**); and a range of 10mm to 20mm for the radius (**R**). For a constraint, you specify a Volume sensor to keep the volume of the water body between 299000mm^3 and 301000mm^3 . For a goal, you use a Mass sensor and specify to minimize the mass of the bottle. The Design Study iterates on the values specified for **L**, **R**, **H**, and Volume, and reports the optimum combination to produce minimum mass.

Day-4
08/02/18

SOLIDWORKS Explorer Overview

SOLIDWORKS Explorer  is a file management tool designed to help you perform such tasks as renaming, replacing, and copying SOLIDWORKS files. You can show a document's references, search for documents using a variety of criteria, and list all the places where a document is used. Renamed files are still available to those documents that reference them.

SOLIDWORKS Explorer Options

You can access the Options dialog box from either the SOLIDWORKS client or SOLIDWORKS Explorer.

Configurations in SOLIDWORKS Explorer

With SOLIDWORKS Explorer, you can list configurations for SOLIDWORKS files.

Hyperlinks

You can list and edit hyperlinks contained in the selected part, assembly, or drawing document.

eDrawings

You can display an eDrawings® image of the selected part, assembly, or drawing document.

SOLIDWORKS Search

SOLIDWORKS Search searches for file names and for text strings in all indexed documents.

Tags

Tags let you associate keywords with documents to make it easier to search for them.

Pack and Go Overview

Gathers all related files for a model design (parts, assemblies, drawings, references, design tables, Design Binder content, decals, appearances, and scenes, and SOLIDWORKS Simulation results) into a folder or zip file.

Pack and Go Dialog Box

The Pack and Go dialog box lists related files to be saved into a folder or zip file.

Rename Document or Replace Document Dialog Box

Using SOLIDWORKS Explorer, you can fix renaming errors.

Compare Utility

The Compare utility compares two documents or two configurations of the same document.

Feature Paint

Feature Paint allows you to copy feature parameters from one feature to others that you select.

Find/Modify Utility

The Find/Modify utility lets you find a set of features in a part that satisfy specified parameter conditions, then edit them in a batch mode.

Find and Replace Annotation

Find and Replace Annotation finds and replaces text for a variety of annotation types in the currently open part, assembly, or drawing document.

Geometry Analysis

Geometry Analysis identifies geometric entities in a part that could cause a problem in other applications. These applications include finite element modeling or computer-aided machining.

Power Select

Power Select allows you to select all the entities (edges, loops, faces, or features) in a part that meet certain criteria that you define.

Report Manager

You can save reports for the following utilities: Compare Features, Compare Geometry, Compare Documents, Compare BOMs, Geometry Analysis, Symmetry Check, and Thickness Analysis. Report Manager is a tool that helps you manage these reports.

Simplify Utility

The Simplify Utility lets you create simplified configurations of a part or assembly to perform analysis.

Symmetry Check Utility

Symmetry Check checks for geometric symmetry in parts about a plane. It identifies symmetrical, asymmetrical, and unique faces.

Feature Parameter vs. Volume-based Feature Simplification

When you apply the Simplify Utility, you can determine features to simplify by comparing feature parameters, or by comparing feature volume.

Thickness Analysis

Use the Thickness Analysis utility to determine different thicknesses of a part. This utility is especially helpful when using thin-walled plastic parts.

Comparison of Sustainability Products

Two products can check the environmental impact of models - SOLIDWORKS® SustainabilityXpress and SOLIDWORKS® Sustainability.

Sustainability Task Pane Views

When you launch Sustainability, it opens in the Task Pane. Use the Sustainability Task Pane to set parameters and get real-time feedback about the environmental impact of designs.

Evaluating the Environmental Impact of a Part

You can use SustainabilityXpress or Sustainability to evaluate a part. Change the materials and processes used to create the part and see the effect in the Environmental Impact dashboard of the Sustainability Task Pane.

Evaluating the Environmental Impact of an Assembly (not available with SustainabilityXpress)

You can use Sustainability to evaluate and improve the environmental impact of assemblies. You can refine an assembly design by adding materials to parts or

components. You can perform a general analysis, based on default values, or a more accurate analysis, by adjusting values for each component.

Adding Materials

You can access additional materials for Sustainability studies using the Sustainability Extras folder in the SOLIDWORKS Materials dialog box.

Visualizing Sustainability Properties

You can use Assembly Visualization to focus on the components that most affect an assembly's environmental impact.

Day-5
09/02/18

SOLIDWORKS MBD (Model Based Definition) lets you create models without the need for drawings giving you an integrated manufacturing solution for the SOLIDWORKS software.

SOLIDWORKS MBD helps companies define, organize, and publish 3D product and manufacturing information (PMI), including 3D model data in industry standard file formats.

- SOLIDWORKS MBD offers 3D PMI definition capabilities using DimXpert and reference dimensions.
- You can use SOLIDWORKS technologies such as annotation views, dynamic viewing of annotation planes and 3D views to organize 3D PMI in a structured, easy-to-locate fashion.
- Besides the native SOLIDWORKS file formats, SOLIDWORKS MBD creates output files such as 3D PDF and eDrawings.

SOLIDWORKS MBD guides the manufacturing process directly in 3D:

- By avoiding the unnecessary and costly revision of 2D drawings, SOLIDWORKS MBD streamlines production, cuts cycle time, and improves communication with the supply chain.
- By using intuitive 3D interaction and rich metadata properties that reduce manufacturing errors.

The SOLIDWORKS MBD add-in:

- Operates within the SOLIDWORKS environment with its own CommandManager.
- Supports all native SOLIDWORKS 3D part and assembly data, such as configurations, constraints, and PMI.

3D Views for Model Based Definition

To support MBD, you can create 3D views of parts and assemblies that contain the model settings needed for review and manufacturing. The output you create lets users navigate back to those settings as they evaluate the design. Similar to 2D drawing views, except these views are in 3D.



KG REDDY

College of Engineering
& Technology

Ref No: KGR CET/ME/2017-18/027

Date: 28/01/2018

CIRCULAR

All the II-Year II-semester B. Tech Mechanical Engineering students are here by informed to enroll for the certification course on “**SOLID WORKS**”, which is conducted by KG Reddy college of Engineering and Technology from 05/02/2018 to 09/02/2018. Interested students are instructed to complete their registration before 03/02/2018.

HOD

Copy to:

1. Exam Section
2. Notice Boards
3. Library

Principal

Principal
KG Reddy College of Engineering & Technology
Chilkur (V), Moinabad (M).
R.R. Dist., Telangana.



KG REDDY COLLEGE OF ENGINEERING & TECHNOLOGY
Chilkur (Vill) Moinabad (Mdl) R R Dist

DEPARTMENT OF MECHANICAL ENGINEERING

**CERTIFICATE COURSE ON SOLID WORKS
SCHEDULE**

Day	Date	Timings	Topic name
1	05/02/18	09:00 to 11:00	Solid Works basics What is Solid Works?
		11:10 to 01:00	File references
		01:45 to 02:50	Opening files, The Solid Works user interface
		02:50 to 04:15	Using the,Command Manager
2	06/02/18	09:00 to 11:00	Introduction to sketching Sketch relations Dimensions Extrude,
		11:10 to 01:00	Sketching guidelines
		01:45 to 02:50	2D sketching, Stages in the process,
		02:50 to 04:15	Basic sketching, Design intent
3	07/02/18	09:00 to 11:00	Basic part modelling Choosing the best profile,
		11:10 to 01:00	Details of the part View options, Filletting, Editing tools,
		01:45 to 02:50	Detailing basics, Drawing views, Centre marks, Dimensioning,
		02:50 to 04:15	Edit material, Mass properties, File properties
4	08/02/18	09:00 to 11:00	Configuration Using drawings Creating configurations, Using configure dimension/feature Using global variables,
		11:10 to 01:00	equations Global variables Equations Modelling strategies for configurations, Editing parts that have configurations
		01:45 to 02:50	Design library, More about making drawings Section, model, broken and detail views,
		02:50 to 04:15	Drawing sheets and sheet formats, Projected views, Annotations
5	09/02/18	09:00 to 11:00	Assembly modelling Creating a new assembly Position of the first component
		11:10 to 01:00	Feature Manager design tree, symbols, Adding components, Subassemblies,



KG REDDY

College of Engineering
& Technology

			Smart mates,
		01:45 to 02:50	Pack and go, Analysing the assembly Checking for clearances
		02:50 to 04:15	Changing the values of dimensions Bill of materials, Assembly drawings



DATE: 05/03/18 TO 09/03/18

[illegible]



KG REDDY

College of Engineering
& Technology

31	16QM5A0307	Chintakuntala Jaisurya	Jai	Jai	Jai	Jai	Jai	Jai	Jai	Jai	Jai	Jai
32	16UR1A0302	K Rajendra	Ra	Ra	Ra	Ra	Ra	Ra	Ra	Ra	Ra	Ra
33	17QM5A0301	Arigela Ranjith	Ranjith	Ranjith	Ranjith	Ranjith	Ranjith	Ranjith	Ranjith	Ranjith	Ranjith	Ranjith
34	17QM5A0302	Barmali Karthik	KB	KB	KB	KB	KB	KB	KB	KB	KB	KB
35	17QM5A0303	Bunne Naveen	Naveen	Naveen	Naveen	Naveen	Naveen	Naveen	Naveen	Naveen	Naveen	Naveen
36	17QM5A0304	Chandrakani Mahesh	CM	CM	CM	CM	CM	CM	CM	CM	CM	CM
37	17QM5A0306	Gajjala Vinod	Vinod	Vinod	Vinod	Vinod	Vinod	Vinod	Vinod	Vinod	Vinod	Vinod
38	17QM5A0307	Golkonda Raj Kumar	RK	RK	RK	RK	RK	RK	RK	RK	RK	RK
39	17QM5A0308	Javoji Hari Krishna	KH	KH	KH	KH	KH	KH	KH	KH	KH	KH
40	17QM5A0309	K Rajashekhar	Raj	Raj	Raj	Raj	Raj	Raj	Raj	Raj	Raj	Raj
41	17QM5A0310	Kanakala Arun Kumar	KAR	KAR	KAR	KAR	KAR	KAR	KAR	KAR	KAR	KAR
42	17QM5A0311	Kokkula Uday	Uday	Uday	Uday	Uday	Uday	Uday	Uday	Uday	Uday	Uday
43	17QM5A0312	Maloth Santhoshkumar	MS	MS	MS	MS	MS	MS	MS	MS	MS	MS
44	17QM5A0313	Mangali Ganesh	Ganesh	Ganesh	Ganesh	Ganesh	Ganesh	Ganesh	Ganesh	Ganesh	Ganesh	Ganesh
45	17QM5A0314	Md Saqlain	Saq	Saq	Saq	Saq	Saq	Saq	Saq	Saq	Saq	Saq
46	17QM5A0315	Mula Ramakanth	MR	MR	MR	MR	MR	MR	MR	MR	MR	MR
47	17QM5A0316	Myagani Krishna Kanth	KK	KK	KK	KK	KK	KK	KK	KK	KK	KK
48	17QM5A0318	Nalapuram Kumar	Kumar	Kumar	Kumar	Kumar	Kumar	Kumar	Kumar	Kumar	Kumar	Kumar
49	17QM5A0320	Panthangi Nikhil	Nikhil	Nikhil	Nikhil	Nikhil	Nikhil	Nikhil	Nikhil	Nikhil	Nikhil	Nikhil
50	17QM5A0322	Rayudu Nagababu	Nagababu	Nagababu	Nagababu	Nagababu	Nagababu	Nagababu	Nagababu	Nagababu	Nagababu	Nagababu
51	17QM5A0323	Sara Vishnuvardhan	Vishnu	Vishnu	Vishnu	Vishnu	Vishnu	Vishnu	Vishnu	Vishnu	Vishnu	Vishnu
52	17QM5A0324	Shaik Luqman	Luqman	Luqman	Luqman	Luqman	Luqman	Luqman	Luqman	Luqman	Luqman	Luqman
53	17QM5A0326	Vulli Shiva Krishna	Shiva	Shiva	Shiva	Shiva	Shiva	Shiva	Shiva	Shiva	Shiva	Shiva

Signature of Coordinator

KG REDDY

CERTIFICATE

Name: AMIRGUDEM MAHESH

Registration No: 16QM1A0303

has successfully completed the prescribed requirements for the award of certificate course on "**SOLID WORKS**" conducted by Mechanical Engineering held in month of February from 05/03/18 to 09/03/18 in the academic year 2017-2018.

Date: 09/03/18



Course Coordinator



Principal



KG REDDY

For the purpose of awarding
the certificate

CERTIFICATE

Name: MATETI AKSHAY KUMAR

Registration No: 14QM1A0386

has successfully completed the prescribed requirements for the award of certificate course on "**SOLID WORKS**" conducted by Mechanical Engineering held in month of February from 05/03/18 to 09/03/18 in the academic year 2017-2018.

Date: 09/03/18

Course Coordinator



Principal