Kandula Srinivasa Reddy Memorial College of Engineering (Autonomous)

Kadapa-516003. AP

(Approved by AICTE, Affiliated to JNTUA, Ananthapuramu, Accredited by NAAC)

(An ISO 9001-2008 Certified Institution)

Department of Mechanical Engineering

Certification Course

on

"SOLID WORKS"

Resource Persons : 1. Sri D. Merwin Rajesh Asst. Professor, Dept. of ME, KSRMCE

Course Coordinators: 1. Smt. E. Reddy Gowthami, Asst. Professor, Dept. of ME, KSRMCE

Date: 05/05/22 to 23/05/22

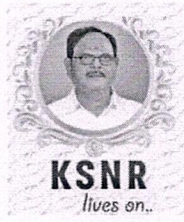
K.S.R.M. COLLEGE OF ENGINEERING

(UGC-AUTONOMOUS)

Kadapa, Andhra Pradesh, India– 516 003

Approved by AICTE, New Delhi & Affiliated to JNTUA, Ananthapuramu.

An ISO 14001:2004 & 9001: 2015 Certified Institution



Lr./KSRMCE/ME/2021-22/ Date: 04-05-2022

To
The Principal,
KSRMCE,
Kadapa.

Sub: Permission to Conduct Certificate Course on "SOLID WORKS" from 05-05-2022 to 23-05-2022 – Reg.

Respected Sir,

The Department of Mechanical Engineering is planning to offer a certification course on "SOLID WORKS" to B. Tech. III Year VI semester students. The course will be conducted from 05-05-2022 to 23-05-2022. In this regard, we are requesting you to grant permission to conduct certificate course.

Thanking you

Forwarded to Principal Sir

Yours faithfully
E Reddy Gowthami
Smt E. Reddy Gowthami, Asst. Professor

Permitted
V. S. S. Murty

K.S.R.M. COLLEGE OF ENGINEERING

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Cr./KSRMCE/ME/2021-22/

Date: 04/05/2022

Circular

The Department of Mechanical Engineering is offering a certification course on "**SOLID WORKS**" From 05-05-2022 to 23-05-2022 to B. Tech III Year VI semester students. In this regard, interested students are required to register for the Certification Course.

The Course Coordinators and Resource Persons
Smt E. Reddy Gowthami, Asst. Professor
Sri. D. Merwin Rajesh, Asst. professor
Dept. of Mechanical Engineering. -KSRMCE.

HOD
Professor & Head
Department of Mechnical Engineering
K.S.R.M. College of Engineering
KADAPA - 516 003.

Cc to:

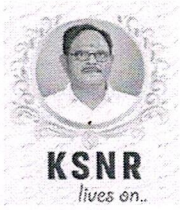
IQAC-KSRMCE

K.S.R.M.COLLEGE OF ENGINEERING

(UGC-AUTONOMOUS)

Kadapa, Andhra Pradesh, India–516 005

Approved by AICTE, NewDelhi & AffiliatedtoJNTUA, Ananthapuramu



Department of Mechanical Engineering

Certification Course on **SOLID WORKS**

List of Participants

S.no	Roll No	Name of the Student	Email Id's
1	189Y1A0364	Khaleel Ahamed	189y1a0364@ksrmce.ac.in
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Coordinator

HOD

Professor & Head
Department of Mechnical Engineering
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KADAPA - 516 003.

K.S.R.M. COLLEGE OF ENGINEERING

(UGC-AUTONOMOUS)

DEPARTMENT OF ,MECHANICAL ENGINEERING

SOLID WORKS

(Certificate Course)

Course objective: To Acquire Basic Knowledge in solid works to Model any Design

Module-1 10 Hrs

Sketching: Outline of course, Overview of the SolidWorks User Interface and suggested settings. Drawing Templates and Sheet Formats : Default Drawing Templates, Sheet Format and Sheet Size, New SolidWorks Document, Sheet Format/Size .Drawing Sheet Size and Format. Draw tools, Editing Tool

Sketch Entities – Inference line, Centerline line, Line, Circle, Arc, Ellipse, Rectangle, Slots, Polygon, Parabola, Ellipse, Partial Ellipse, Spline, Spline tools, Spline on surface, Equation driven curve, Points, Text, Construction geometry, Snap, grid,

Sketch Tools - Fillet, Chamfer, Offset, Convert entities, Intersection curve, Face curve, Trim, Extend, Split, Jog Line, Construction Geometry, Mirror, Dynamic Mirror, Move, Copy, Rotate, Scale, Stretch, Sketch pattern , Polygon, Make path, Close Sketch To Model, Sketch picture, Check Sketch for Feature, Area hatch/Fill

Dimensioning - Smart, Horizontal, Vertical, Ordinate, Horizontal ordinate, Vertical ordinate, Align ordinate, Fully define sketch. Sketch Diagnosis, Sketch Xpert, 3D Sketching, Rapid Sketch

Exercises

Module-2 10Hrs

Part Modelling

Part Modeling Tools Creating reference planes Creating Extrude features – Direction1, Direction2, From option, Thin feature, Applying draft, Selecting contours Creating Revolve features – Selecting Axis, Thin features, Selecting contours Creating Swept features-Selecting, Profile and Path, Orientation/twist type, Path Alignment, Guide Curves, Start/End tangency, Thin feature

Creating Loft features – Selecting Profiles, Guide curves, Start/End Constraints, Centerline parameters, Sketch tools, Close loft. Selecting geometries – Selection Manager, Multiple Body concepts Creating Reference - points, axis, coordinates

Creating Chamfer Creating Shell Creating Rib Creating Pattern - Linear pattern, Circular pattern, Sketch driven pattern, Curve driven pattern, Table driven pattern, Fill pattern, mirror

Exercises

Module-3 10Hrs

Assembly

Assembly Modeling Tools Introduction to Assembly Modeling & Approaches – Top down and Bottom up approach Applying Standard Mates- Coincident, Parallel, Perpendicular, Tangent, Concentric, Lock, Distance, Angle.

Applying Advanced Mates – Symmetric, Width, Path Mate, Linear/Linear Coupler, Limit Mate. Applying Smart mates Applying Mate reference

Manipulating Components - Replacing Components, Rotating Components, Move Components, Collision Detection, Physical Dynamics, Dynamic Clearance, Detecting Interference Creating Pattern - Assembly Pattern, Mirror Creating Explode Views Top Down Design – Layout Sketch, Work Part In the Context of an assembly. Smart Components, Smart Fasteners, Physical Simulation

Exercises

Course Outcomes:

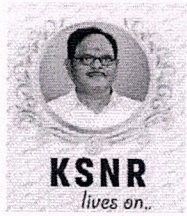
After Successful completion of the course the students will be able to

CO1. Demonstrate competency with multiple drawing and modification commands in SolidWorks.

CO2. Create three-dimensional solid models.

CO3. Create three-dimensional assemblies incorporating multiple solid models.

CO4. Apply industry standards in the preparation of technical mechanical drawings.



SCHEDULE

Department of Mechanical Engineering

Certification course
on
"SOLID WORKS"

Date	Timing	Course Instructor	Topic to be covered
05/05/2022	4 PM to 6 PM	D.MERWIN RAJESH	INTRODUCTION TO SOLID WORKS SKETCHING: Outline of course, Overview of the Solid Works User Interface and suggested settings.
06/05/2022	4 PM to 6 PM	E.REDDY GOWTHAMI	Drawing Templates and Sheet Formats : Default Drawing Templates, Sheet Format and Sheet Size, New Solid Works Document, Sheet Format/Size .Drawing Sheet Size and Format. Draw tools, Editing Tool
08/05/2022	4 PM to 6 PM	D.MERWIN RAJESH	Sketch Entities – Inference line, Centerline line, Line, Circle, Arc, Ellipse, Rectangle, Slots, Polygon, Parabola, Ellipse, Partial Ellipse, Spline, Spline tools, Spline on surface, Equation driven curve, Points, Text, Construction geometry, Snap, grid,
09/05/2022	4 PM to 6 PM	D.MERWIN RAJESH	Sketch Tools - Fillet, Chamfer, Offset, Convert entities, Intersection curve, Face curve, Trim, Extend, Split, Jog Line, Construction Geometry, Mirror, Dynamic Mirror, Move, Copy, Rotate, Scale, Stretch, Sketch pattern , Polygon, Make path, Close Sketch To Model, Sketch picture, Check Sketch for Feature, Area hatch/Fill
10/05/2022	4 PM to 6 PM	D.MERWIN RAJESH	Dimensioning - Smart, Horizontal, Vertical, Ordinate, Horizontal ordinate, Vertical ordinate, Align ordinate, Fully define sketch. Sketch Diagnosis, Sketch expert, 3D Sketching, Rapid Sketch and EXERCISES
11/05/2022	4 PM to 6 PM	D.MERWIN RAJESH	Part Modeling: Part Modeling Tools Creating reference planes Creating Extrude features – Direction1, Direction2, From option, Thin

			feature, Applying draft, Selecting contours
12/05/2022	4 PM to 6 PM	D.MERWIN RAJESH	Creating Revolve features – Selecting Axis, Thin features, Selecting contours Creating Swept features-Selecting, Profile and Path,
13/05/2022	4 PM to 6 PM	D.MERWIN RAJESH	Orientation/twist type, Path Alignment, Guide Curves, Start/End tangency, Thin feature
16/05/2022	4 PM to 6 PM	D.MERWIN RAJESH	Creating Loft features – Selecting Profiles, Guide curves, Start/End Constraints, Centerline parameters, Sketch tools, Close loft. Selecting geometries – Selection Manager, Multiple Body concepts Creating Reference - points, axis, coordinates
17/05/2022	4 PM to 6 PM	E.REDDY GOWTHAMI	Creating Chamfer Creating Shell Creating Rib Creating Pattern - Linear pattern, Circular pattern, Sketch driven pattern, Curve driven pattern, Table driven pattern, Fill pattern, mirror EXERCISES
18/05/2022	4 PM to 6 PM	D.MERWIN RAJESH	Assembly Modeling Tools Introduction to Assembly Modeling & Approaches – Top down and Bottom up approach Applying Standard Mates- Coincident, Parallel, Perpendicular, Tangent, Concentric, Lock, Distance, Angle
19/05/2022	4 PM to 6 PM	D.MERWIN RAJESH	Applying Advanced Mates – Symmetric, Width, Path Mate, Linear/Linear Coupler, Limit Mate. Applying Smart mates Applying Mate reference
20/05/2022	4 PM to 6 PM	D.MERWIN RAJESH	Manipulating Components - Replacing Components, Rotating Components, Move Components, Collision Detection, Physical Dynamics, Dynamic Clearance, Detecting Interference Creating Pattern - Assembly Pattern,
21/05/2022	4 PM to 6 PM	D.MERWIN RAJESH	Mirror Creating Explode Views Top Down Design – Layout Sketch, Work Part In the Context of an assembly. Smart Components, Smart Fasteners, Physical Simulation
23/05/2022	4 PM to 6 PM	D.MERWIN RAJESH	Exercises and Conclusion

HOD
Professor & Head
Department of Mechnical Engineering
K.S.R.M. College of Engineering
KADAPA - 516 003.

Resource persons: D. Merwin Rajesh

Coordinator: E. Reddy Gowthami,

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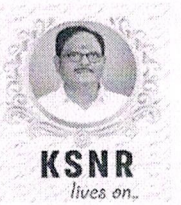


Department of Mechanical Engineering

Certification Course on **Solid Works**

Attendance Sheet

S. No	Name of the Student	Roll List	05/05	06/05	07/05	09/05	10/05	11/05	12/05	13/05	16/05	17/05	18/05	19/05	20/05	21/05	23/05
1	KHALEEL AHAMED	189Y1A0364	P	P	P	A	P	P	P	P	P	P	P	P	P	P	P
2	BIJLI SATISH KUMAR	199Y5A0304	P	P	P	P	P	A	P	P	P	P	P	P	P	P	P
3	SIVARAM NAIK KETHAVATH	199y5a0315	P	P	P	P	P	A	P	P	P	P	P	P	P	P	P
4	VARIKUNTA MUNI DINESH PRAMOD RAJU	199Y5A340	P	A	P	P	P	P	P	P	P	P	P	P	P	P	P
5	MITAIABDULSHEIK MOHAMMED BASHA	199Y5A0326	P	P	P	P	P	P	P	P	P	A	P	P	P	A	P
6	VENNAPUSA UMESH CHANDRA REDDY	199Y5A0342	P	P	P	P	P	P	P	P	P	P	A	P	A	P	P
7	YADAVAKUNTA SIVA RAMI REDDY	199Y5A0343	P	P	P	P	P	P	A	P	P	P	P	A	P	P	P
8	VENNAPUSA SREEKANTH REDDY	199Y5A0341	A	P	P	P	P	P	P	P	P	P	P	P	P	P	P
9	YEDDULA GANGA PRASAD REDDY	199Y5A0344	P	P	P	P	P	P	P	P	P	P	A	P	P	P	P
10	PICHIPATI SHAIK MAHAMMAD AFRID	199Y5A0331	P	P	P	P	P	A	P	P	P	P	P	P	P	P	P
11	SHAIK GHOUSE LAZAM	199Y5A0335	P	P	P	P	P	P	P	P	P	P	P	P	P	P	P
12	YEDDULA PRAVEEN KUMAR	199Y5A0345	P	A	P	P	P	P	P	P	P	P	P	P	P	P	P

	REDDY																
13	CHANDOLI SREENIVASULU	199Y5A0307	P	A	P	P	P	P	P	P	P	P	P	P	P	P	P
14	GAJJALA VEERA PRASAD	199Y5A0309	P	P	A	P	P	P	P	P	P	P	P	P	P	P	P
15	CHEPURI AMARENDRA KUMAR	199Y5A0308	P	P	P	P	P	P	A	P	P	P	P	P	P	P	P
16	Y BHANU PRAKASH	199y5a0347	P	A	P	P	P	P	P	P	P	P	P	P	P	P	P
17	SHAIK SHEKSHAVALI	199Y5A0338	P	A	P	P	P	P	P	P	A	P	P	P	P	P	P
18	N HARI KRISHNA	199Y5A0328	P	P	P	P	P	P	P	P	P	P	P	P	P	P	P
19	N. HARI OBULESU	199Y5A0329	P	P	P	P	P	P	A	P	P	P	P	P	P	P	P
20	MALLELA RAJASHEKHAR REDDY	199Y5A0325	A	P	P	P	P	P	P	P	P	P	P	P	P	P	P
21	SHAIK ZUBAIR	189y1a0362	P	P	P	P	P	P	P	P	A	P	P	P	P	P	P
22	THAMBALA VEERESH	189Y1A0370	P	P	P	P	P	P	P	P	P	P	P	P	P	P	P
23	THIRUVAIPATI SASIKANTH	189y1a0371	P	P	P	P	P	P	P	P	P	P	P	A	P	P	P
24	SIDDAREDDY LINGAMAIAH	189Y1A0366	P	P	P	P	P	P	P	P	P	P	P	P	P	P	P
25	VUKKADAM MAHESH KUMAR	189y1a0373	P	P	P	P	P	P	P	P	P	P	P	P	P	P	P
26	VADDEMANI LOKESHWAR REDDY	189y1a0374	P	P	P	P	P	P	P	P	P	P	P	P	P	P	P
27	S.GHAYAZ AHMED	189y1a0368	P	P	A	P	P	P	P	P	P	P	P	P	A	P	P
28	SHAIK ZUBAIR HUSSAIN	189Y1A0363	P	P	P	A	P	P	P	P	P	P	P	P	P	P	P
29	THAMBALA VEERESH	189Y1A0370	P	P	P	P	P	P	P	P	A	P	P	P	P	P	P
30	SHARON SAMUEL	189Y1A0365	P	P	P	P	P	P	P	P	P	P	P	P	P	P	P
31	SHARON SAMUEL	189Y1A0365	P	P	P	P	P	P	P	P	P	P	P	P	P	P	P
32	ATHMAKURU MAHESH BABU	199y5a0303	P	P	P	P	P	P	P	P	P	A	P	P	P	P	P
33	SHAIK IMRAN	199Y5A0336	P	P	P	P	P	P	P	P	P	P	P	P	P	P	P
34	G. VAMSI	199Y5A0310	A	P	P	P	P	P	P	P	P	P	P	A	P	P	P
35	JINKALA SUBHAN	199Y5A0312	P	P	P	P	A	P	P	A	P	P	A	P	P	A	P
36	PUTTA SASIKANTH REDDY	199Y5A0332	P	P	P	P	P	A	P	P	A	P	P	P	P	P	P
37	SHAIK IMRAN	199Y5A0336	P	P	A	P	P	P	P	P	P	P	P	P	P	P	P
38	KONETI VENKATA SIVA MANORANJAN	199Y5A0317	P	P	P	P	P	P	P	P	P	P	P	P	P	P	P
39	S. VENKATESH	189Y1A0367	P	P	P	P	P	P	P	P	A	P	A	P	P	P	A
40	MOOLA.ACHYUTH REDDY	199Y5A0327	P	A	P	P	P	P	P	P	P	P	P	P	P	P	P

41	KONDURU VENKATESH	199Y5A0316	A	P	P	P	P	P	P	P	P	P	P	P	P	P	P	P
42	LAKSHMI NARASIMHA	199Y5A0324	P	P	P	A	P	P	P	P	P	P	P	P	P	P	P	P
43	ALAMURU MABU BASHA	199Y5A0301	P	A	P	P	P	A	P	P	P	P	P	A	P	P	P	P
44	MADDURU SAITEJA	199Y5A0321	P	P	P	P	A	P	P	P	P	P	P	P	P	P	P	P
45	KOTHAPALLI PRUDHVI	199Y5A0318	P	P	P	P	P	P	P	P	P	P	P	P	P	P	P	P
46	SANNA GURAPPA	199Y5A0334	A	P	P	A	P	P	P	P	P	P	P	P	P	P	P	P
47	Y ANDERSON	189Y1A0376	P	P	P	A	P	P	P	P	P	P	P	P	P	P	P	P
48	YERRAMREDDY CHENNA KESAVA REDDY	189Y1A0377	P	P	P	A	P	P	P	P	P	P	P	P	P	P	P	P
49	KURUVA KUMAR	199Y5A0320	P	P	P	P	P	P	P	P	P	P	P	P	P	P	P	P
50	BODAGALA SAIBHARATH	199Y5A0305	P	P	P	P	P	P	P	P	P	P	P	P	P	P	P	P
51	KURUVA BHASKAR	199y5a0319	P	P	P	P	P	P	P	P	P	P	P	P	P	P	P	P
52	ANAND REDDY	199Y5A0333	P	P	P	P	P	P	P	P	P	P	P	P	P	P	P	P
53	MAJJARI CHARAN	199y5a0322	P	P	P	P	P	P	P	P	P	P	P	P	P	P	P	P
54	BOGGULA OBULA REDDY	199y5a0306	P	P	P	P	P	P	P	P	P	P	P	P	P	P	P	P
55	YERRAGORLA BRAMHAIAH	199Y5A0346	P	P	P	P	P	P	P	P	P	P	P	P	P	P	P	P
56	THUMMALA MOHAMMED HAROON	189y1a0372	P	P	P	P	P	P	P	P	P	P	P	P	P	P	P	P
57	URUMU SUDHEER KUMAR	199Y5A0339	P	P	P	P	P	P	P	P	P	P	P	P	P	P	P	P
58	SHAIK MAHABOOB BASHA	199Y5A0337	P	P	P	P	P	P	P	P	P	P	P	P	P	P	P	P

E. P. Gowtham
Coordinator

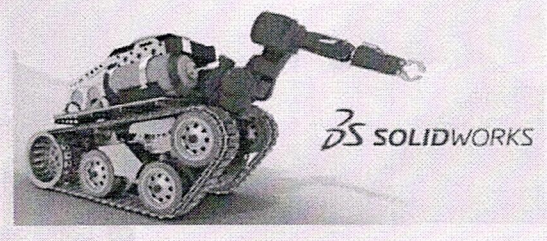
HOD

Professor & Head
Department of Mechnical Engineering
K.S.R.M. College of Engineering
KADAPA - 516 003.

K.S.R.M. COLLEGE OF ENGINEERING

(UGC - Autonomous)
Kadapa, Andhra Pradesh, India– 516 003
Approved by AICTE, New Delhi & Affiliated to JNTUA, Ananthapuramu.
Department of Mechanical Engineering

CERTIFICATE COURSE ON
SOLIDWORKS



DATE OF EVENTS

05/05/2022 to 23/05/2022
FROM
4.00 pm to 6.00 pm

Course Instructors

Smt . E.ReddyGowthami
Asst.Professor,MED
Sri.D.Merwin Rajesh
Asst.Professor,MED

VENUE: ENGINEERING GRAPHICS AND DESIGN LAB, ME DEPARTMENT

DR.D.RAVIKANTH (HOD) | DR. V.S.S. MURTHY (Principal) | PROF. A. MOHAN (Director) | DR. KANDULA CHANDRA OBUL REDDY (Managing Director) | SMT. K. RAJESWARI (Correspondent, Secretary, Treasurer) | SRI K. MADAN MOHAN REDDY (Vice - Chairman) | SRI K. RAJA MOHAN REDDY (Chairman)

Ksrmceofficial | www.ksrmce.ac.in | 8143731980, 8575697569

Report of
Certification Course on "SOLID WORKS"
From 05/05/2022 to 23/05/2022

Target Group	:	**VI semester Students**
Details of Participants	:	**58 Students**
Co-coordinator(s)	:	**Smt. E. Reddy Gowthami**
Resource Persons	:	**Sri D. Merwin Rajesh**
Organizing Department	:	**Mechanical Engineering**
Venue	:	**Engineering Graphics Lab, Mechanical Department**

Description:

The Department of Mechanical Engineering conducted a certification course on "SOLID WORKS" from 5th May 2022 to 23rd May 2022. The course duration was 30 hours .The course Resource Persons are Sri P.Siva Seshu, Assistant Professor Department Mechanical Engineering, KSRMCE.

SOLIDWORKS is an easy to use parametric design modular, meaning you can easily edit the design at any stage in the design process. Real View graphics allow you to visualize your design in real time whilst Photo View 360 can create sophisticated photo realistic renderings and animations. Both tools will give you a fantastic insight into the way your design will look without it actually being made and can be a powerful asset when presenting your work to customers. You can look at each individual part of the design, see accurate mass properties and check for interference, meaning that you won't have to build/manufacture the product before you see any errors, saving time and money and reducing the number of prototypes needed. All of this will speed up the whole process of design as you know it and increase productivity.

" SOLIDWORKS is the preeminent software for computer-aided design and computer-aided engineering. Multiple parts can be combined together into assemblies by assigning relationships (called "mates") among various features. Both assemblies and individual parts can be turned into engineering drawings where things like dimensions, notes, and revision numbers are typically catalogued. The software is basically a one-stop shop for design so that you can turn ideas into reality. "

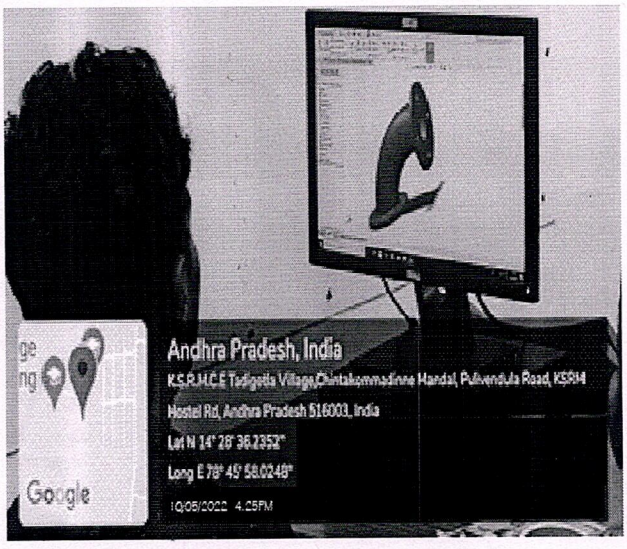
At the End of the course Students will able to do 2D modeling, Part Modeling and Assembly of components and able to Model Industry standard Mechanical Drawings.

Photos

The pictures taken during the course are given below:

Photos

The pictures taken during the course are given below:



Photos: When Students were Practicing the Solid Works

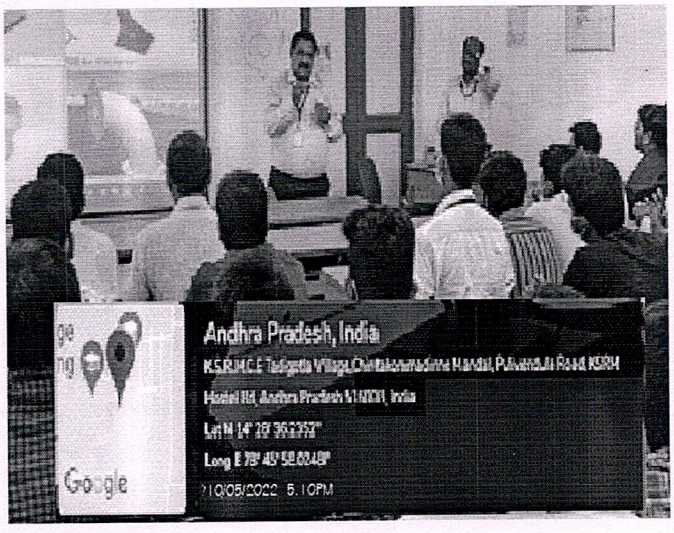


Photo : Motivating the Students

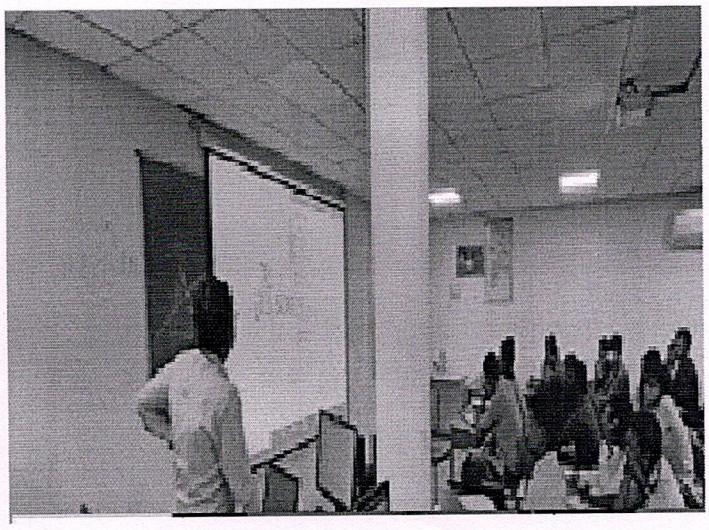


Photo: Resource person Explaining about the Course

Coordinators

HOD

Professor & Head
Department of Mechnical Engineering
K.S.R.M. College of Engineering
KADAPA - 516 003.

K.S.R.M. COLLEGE OF ENGINEERING

(UGC - Autonomous)

Kadapa, Andhra Pradesh, India– 516 003

Approved by AICTE, New Delhi & Affiliated to JNTUA, Ananthapuramu.

KSNR
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Certificate of Completion

This to certify that Mr/Mrs. N. HARI KRISHNA Bearing the Roll Number 199Y5A0328 has Successfully Completed Certification Course on " SOLID WORKS ", from 05/05/2022 to 23/05/2022, Organized by Department of Mechanical Engineering, KSRMCE, Kadapa.

E. P. Gowtham
Coordinator

HOD ME
Professor & Head
Department of Mechnical Engineering
K.S.R.M. College of Engineering
KADAPA - 516 003.

V. S. S. Murty
Principal
PRINCIPAL
K.S.R.M. COLLEGE OF ENGINEERING
KADAPA - 516 003. (A.P.)

K.S.R.M. COLLEGE OF ENGINEERING

(UGC - Autonomous)

Kadapa, Andhra Pradesh, India– 516 003

Approved by AICTE, New Delhi & Affiliated to JNTUA, Ananthapuramu.

KSNR
lives on..

Certificate of Completion

This to certify that Mr/Mrs. K. PRUDHVI Bearing the Roll Number 189Y1A0318 has Successfully Completed Certification Course on " SOLID WORKS "

from 05/05/2022 to 23/05/2022, Organized by Department of Mechanical Engineering, KSRMCE, Kadapa.

K.P. Goud
Coordinator

HOD ME

Professor R Roel
Department of Mechanical Engineering
K.S.R.M. College of Engineering

V.S.S. Murthy
Principal

PRINCIPAL
K.S.R.M. COLLEGE OF ENGINEERING
KADAPA - 516 003 (A.P.)

K.S.R.M. COLLEGE OF ENGINEERING

(UGC - Autonomous)

Kadapa, Andhra Pradesh, India– 516 003

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KSNR
lives on..

Certificate of Completion

This to certify that Mr/Mrs. KHALEEL AHMED Bearing the Roll Number 189Y1A0364 has Successfully Completed Certification Course on " SOLID WORKS "

from 05/05/2022 to 23/05/2022, Organized by Department of Mechanical Engineering, KSRMCE, Kadapa.

E.R. Gowthami
Coordinator

HOD ME
Professor & Head
Department of Mechnical Engineering
K.S.R.M. College of Engineering
KADAPA - 516 003.

V. S. S. Murty
Principal
PRINCIPAL
K.S.R.M. COLLEGE OF ENGINEERING
KADAPA - 516 003. (A.P.)

Feedback on Certificate Course on "Solid works" From 05/05/2022 to 23/05/2022

*Required

1. Student Name (Optional)

2. Roll Number (Optional)

3. The objectives of the course were met (Objective) *

Mark only one oval.

- Excellent
- Good
- Satisfactory
- Poor

4. The pace of the course was appropriate to the content and attendees(Content) *

Mark only one oval.

- Excellent
- Good
- Satisfactory
- Poor

5. The content of the course was organized and easy to follow (Delivery) *

Mark only one oval.

- Excellent
- Good
- Satisfactory
- Poor

6. The Resource Persons were well prepared and able to answer any questions (Interaction) *

Mark only one oval.

- Excellent
- Good
- Satisfactory
- Poor

7. The exercises / role play were helpful and relevant (Syllabus Coverage) *

Mark only one oval.

- Excellent
- Good
- Satisfactory
- Poor

8. The venue was appropriate for the course (About Venue)*

Mark only one oval.

- Excellent
- Good
- Satisfactory
- Poor

9. The Course satisfy my expectation as a value added Programme (Course Satisfaction) *

Mark only one oval.

- Excellent
- Good
- Satisfactory
- Poor

10. Any Other comments

Feedback on Certificate Course on" Solid Works " from 05/05/22 to 23/05/22

S.No	Timestamp	The objec	The pa	The content	The Reso	The exercisces	The venu	The Cour	Student Nam	Roll Number(C	Any Other comments
1	24/05/2022 16:30:36	Excellen	Excelle	Excellent	Excellent	Excellent	Excellent	Excellent			
2	24/05/2022 16:30:42	Excellent	Excelle	Excellent	Excellent	Excellent	Excellent	Excellent	K Karthik	209y1a0323	
3	24/05/2022 16:30:48	Excellent	Good	Excellent	Excellent	Good	Excellent	Good			
4	24/05/2022 16:30:52	Good	Good	Good	Good	Good	Good	Good	Lokeshwar	209y1a0335	--
5	24/05/2022 16:30:55	Excellent	Excelle	Excellent	Good	Excellent	Excellent	Excellent			
6	24/07/2022 16:30:59	Excellent	Excellent	Excellent	Excellent	Excellent	Excellent	Excellent			
7	24/07/2022 16:31:06	Excellent	Excelle	Excellent	Excellent	Excellent	Good	Good			We need some more EXAMPLES
8	24/05/2022 16:31:10	Excellent	Excelle	Excellent	Good	Excellent	Good	Good			
9	24/05/2022 16:31:13	Satisfacto	Satisfactory	Satisfactory	Satisfactory	Satisfactor y	Satisfactory	Satisfactory	SIDDHAMSE	209y1a0358	No
10	24/05/2022 16:31:23	Good	Excellent	Good	Excellent	Good	Excellent	Excellent			
11	24/05/2022 16:31:27	Excellent	Excellent	Excellent	Excellent	Excellent	Good	Good			
12	24/05/2022 16:31:32	Excellent	Good	Good	Excellent	Good	Good	Excellent			
13	24/05/2022 16:31:36	Good	Good	Good	Good	Good	Satisfacto	Satisfacto			
14	24/05/2022 16:31:42	Good	Good	Good	Good	Good	Good	Good	Shaik Moula	355	It is useful for us
15	24/05/2022 16:31:46	Good	Good	Good	Good	Good	Good	Good			--
16	24/05/2022 16:31:55	Excellent	Excellent	Excellent	Excellent	Excellent	Excellent	Excellent	R.valeedh		
17	24/05/2022 16:32:01	Excellent	Excellent	Excellent	Excellent	Excellent	Excellent	Excellent	Narayana. p		
18	24/05/2022 16:32:03	Excellent	Good	Excellent	Good	Excellent	Excellent	Good			i Learned something n
19	24/05/2022 16:32:07	Excellent	Excellent	Excellent	Excellent	Good	Good	Good	Sagabala Bh	209y1a0351	
20	24/05/2022 16:32:16	Good	Good	Good	Excellent	Good	Good	Excellent	Shaik jubair	219y5a0333	
21	24/05/2022 16:32:21	Excellent	Excellent	Good	Excellent	Good	Excellent	Good			
22	24/05/2022 16:32:25	Good	Good	Good	Good	Good	Good	Good			--
23	24/05/2022 16:32:29	Excellent	Good	Good	Satisfacto	Excellent	Excellent	Excellent			
24	24/05/2022 16:32:36	Excellent	Excellent	Excellent	Excellent	Excellent	Excellent	Excellent			-
25	24/05/2022 16:32:28	Excellent	Excellent	Excellent	Excellent	Excellent	Excellent	Excellent	VELLATUR A	209Y1A0364	Nothing to say
26	24/05/2022 16:32:41	Good	Satisfactor y	Excellent	Excellent	Excellent	Satisfacto	Satisfactory	Sagabala bh	209y1a0351	
27	24/05/2022 16:32:48	Good	Excellent	Excellent	Good	Excellent	Good	Excellent			
28	24/05/2022 16:32:54	Good	Good	Good	Good	Good	Good	Good			--
29	24/05/2022 16:33:06	Excellent	Excellent	Excellent	Excellent	Excellent	Excellent	Excellent			
30	24/05/2022 16:33:16	Excellent	Excellent	Excellent	Excellent	Excellent	Excellent	Excellent			
31	24/05/2022 16:33:21	Excellent	Excellent	Good	Good	Good	Excellent	Good			
32	24/05/2022 16:33:36	Good	Good	Excellent	Good	Good	Good	Good	GALLA MUK	219y5A0311	

									MUK		
33	24/05/2022 16:34:48	Excellent	Excelle	Good	Good	Good	Good	Good			
34	24/05/2022 16:34:54	Excellent	Excelle	Excellent	Excellent	Excellent	Excellent	Excellent			-
35	24/05/2022 16:34:56	Good	Good	Good	Good	Good	Good	Good	SIDDHAM SE	209y1a0358	No
36	24/05/2022 16:34:59	Excellent	Good	Good	Good	Satisfactor	Excellent	Excellent			
37	24/05/2022 16:35:06	Good	Good	Good	Good	Good	Good	Good			No
38	24/05/2022 16:35:13	Excellent	Excelle	Good	Good	Good	Good	Good			
39	24/05/2022 16:35:21	Excellent	Excelle	Excellent	Excellent	Excellent	Excellent	Excellent			
40	24/05/2022 16:35:26	Good	Good	Good	Good	Good	Good	Good			
41	24/05/2022 16:35:31	Good	Good	Good	Good	Good	Good	Good			
42	24/05/2022 16:35:36	Excellent	Excelle	Excellent	Excellent	Excellent	Excellent	Excellent	Korivi venkat	209y1a0327	
43	24/05/2022 16:35:41	Good	Good	Good	Good	Good	Good	Good			
44	24/05/2022 16:35:45	Good	Good	Good	Good	Good	Good	Good			--
45	24/05/2022 16:35:51	Excellent	Excelle	Excellent	Excellent	Excellent	Excellent	Excellent			
46	24/05/2022 16:35:54	Good	Good	Good	Good	Good	Good	Good	V Bharath	219Y5A0341	
47	24/05/2022 16:35:58	Excellent	Excelle	Good	Good	Excellent	Excellent	Good			Nothing
48	24/05/2022 16:36:05	Good	Good	Good	Good	Good	Good	Good			
49	24/05/2022 16:36:11	Excellent	Good	Excellent	Good	Good	Excellent	Excellent	Sagabala Bh	209y1a0351	
50	24/05/2022 16:36:21	Excellent	Excelle	Excellent	Excellent	Excellent	Excellent	Excellent			
51	24/05/2022 16:36:25	Good	Excelle	Good	Good	Good	Good	Good	Shaik Moula	355	Ok
52	24/05/2022 16:36:36	Excellent	Excelle	Excellent	Excellent	Excellent	Excellent	Excellent			
53	24/05/2022 16:36:45	Excellent	Excelle	Good	Good	Good	Good	Good			
54	24/05/2022 16:37:54	Good	Good	Excellent	Good	Excellent	Excellent	Excellent			
55	24/05/2022 16:38:06	Excellent	Excelle	Excellent	Excellent	Excellent	Excellent	Excellent			
56	24/05/2022 16:38:16	Good	Excelle	Excellent	Good	Good	Good	Good			
57	24/05/2022 16:38:22	Excellent	Excelle	Excellent	Excellent	Excellent	Excellent	Excellent			
58	24/05/2022 16:38:36	Excellent	Excelle	Good	Good	Good	Good	Good	P obula vam	209Y1A0346	

Coordinator

HOD

K.S.R.M. COLLEGE OF ENGINEERING (AUTONOMOUS), KADAPA-516003
DEPARTMENT OF MECHANICAL ENGINEERING
VALUE ADDED COURSE ON
SOLID WORKS
FROM 05/05/2022 TO 23/05/2022
AWARD LIST

S.No	Roll Number	Name of the Student	Marks Obtained
1	189Y1A0362	Shaik Zubair	14
2.	189Y1A0363	Shaik Zubair Hussain	12
3.	189Y1A0364	Shaik Khaleel Ahamed	13
4.	189Y1A0365	Sharon Samuel	13
5.	189Y1A0366	Siddareddy Lingamaiah	14
6.	189Y1A0367	Koneti Venkata Siva Manoranjan	12
7.	189Y1A0368	S.Ghayaz Ahmed	13
8.	189Y1A0370	Thambala Veeresh	14
9.	189Y1A0371	Thiruvaipati Sasikanth	12
10	189Y1A0372	T. Mohammed	12
11	189Y1A0373	Vukkadam Mahesh Kumar	14
12	189Y1A0374	Vaddemani Lokeshwar Reddy	13
13	189Y1A0376	Y Anderson	13
14	189Y1A0377	Yerramreddy Chenna Kesava Reddy	12
15	199Y5A0301	Alamuru Mabu Basha	14
16	199Y5A0303	Athmakuru Mahesh Babu	12
17	199Y5A0304	Bijli Satish Kumar	12
18	199Y5A0305	Bodagala Saibharath	13
19	199Y5A0306	Boggulla Obulreddy	14
20	199Y5A0307	Chandoli Sreenivasulu	12
21	199Y5A0308	Chepuri Amarendra Kumar	14
22	199Y5A0309	Gajjala Veera Prasad	13
23	199Y5A0310	G. Vamsi	12
24	199Y5A0312	Jinkala Subhan	14
25	199Y5A0315	Sivaram Naik Kethavath	12
26	199Y5A0316	Konduru Venkatesh	13
27	199Y5A0317	Koneti Venkata Siva Manoranjan	14
28	199Y5A0318	Kothapalli Prudhvi	12
29	199Y5A0319	Kuruva Bhaskar	13
30	199Y5A0320	Kuruva Kumar	14

31	199Y5A0321	Madduru Saiteja	12
32	199Y5A0322	Majjari Charan	14
33	199Y5A0324	Lakshmi Narasimha	13
34	199Y5A0325	Mallela Rajashekhar Reddy	12
35	199Y5A0326	Mitaiabdulsheik Mohammed Basha	12
36	199Y5A0327	Moola.Achyuth Reddy	13
37	199Y5A0328	N Hari Krishna	14
38	199Y5A0329	N. Hari Obulesu	12
39	199Y5A0330	U.Sudheer Kumar	14
40	199Y5A0331	Pichipati Shaik Mahammad Afrid	12
41	199Y5A0332	Putta Sasikanth Reddy	13
42	199Y5A0333	Anand Reddy	14
43	199Y5A0334	Sanna Gurappa	12
44	199Y5A0335	Shaik Ghouse Lazam	14
45	199Y5A0336	Shaik Imran	13
46	199Y5A0337	Shaik Mohaboob Basha	14
47	199Y5A0338	Shaik Shekshavali	12
48	199Y5A0340	Varikunta Muni Dinesh Pramod Raju	14
49	199Y5A0341	Vennapusa Sreekanth Reddy	12
50	199Y5A0342	Vennapusa Umesh Chandra Reddy	12
51	199Y5A0343	Yadavakunta Siva Rami Reddy	13
52	199Y5A0344	Yeddula Ganga Prasad Reddy	14
53	199Y5A0345	Yeddula Praveen KumarReddy	12
54	199Y5A0346	Y. Bramiah	14
55	199Y5A0347	Y Bhanu Prakash	13

Coordinator

HoD

Professor & Head
Department of Mechnical Engineering
K.S.R.M. College of Engineering
KADAPA - 516 003.

K.S.R.M. COLLEGE OF ENGINEERING (AUTONOMOUS), KADAPA-516003
DEPARTMENT OF MECHANICAL ENGINEERING
VALUE ADDED /CERTIFICATE COURSE ON
SOLID WORKS
05/05/2022 TO 23/05/2022
ASSESSMENT TEST

14

Roll Number: 18Y1A0370 **Name of the Student:** T. Veeresh

Time: 20 Min **(Objective Questions)** **Max.Marks:** 20

Note: Answer the following Questions and each question carries **one** mark.

1.What is the purpose of the Solid Works User Interface? [C]

a) To create and edit document b) To manage files and folders

c) To browse the internet d) to play games

2. Which section of the Solid Works interface is typically used for creating and [b]

Modifying 3D models?

a) Design Library

b) Feature Manager Design Tree

c) Task Pane

d) Command Manager

3. What is the purpose of a drawing template in Solid Works? [a]

a) To store sketch entities

b) To define the layout of a drawing sheet

c) To create 3D models

d) To generate G-code for CNC machines

4. What is the purpose of the Fillet tool in sketching? [a]

a) To add text annotations b) To create rounded corners

c) To draw straight lines d) To insert images

5. What is the primary function of the Solid Works User Interface? [C]

a) Gaming b) Browsing the internet

c) Creating and editing documents d) Managing emails

6. Which area of the Solid Works interface is used for managing feature history and components? [b]

a) Task Pane
b) Feature Manager Design Tree
c) Design Library
d) Command Manager

7. How can you customize Solid Works to better suit your preferences? [d]

a) By changing the default font
b) By adjusting settings in the Solid Works Options
c) By installing additional plugins
d) By updating your operating system

8. Which sketch tool is used to create a rounded internal corner between two lines Or curves? [a]

a) Fillet
b) Chamfer
c) Offset
d) Convert entities

9. The Offset tool in Solid Works sketching is primarily used to create: [c]

a) Copies of sketch entities
b) Symmetrical sketches
c) A parallel copy of a sketch entity
d) Tangent curves

10. The Intersection Curve tool in Solid Works sketching helps create a curve at The intersection of: [d]

a) Two planar faces
b) Two parallel lines
c) A line and an arc
d) A sketch and a reference plane

11. The "Jog Line" tool in Solid Works sketching is used to: [a]

a) Create zigzag patterns
b) Add symbols to the sketch

c) Generate text

d) Offset a portion of a line

12. What are the two main approaches to assembly modeling in Solid Works? [b]

a) Standard and Advanced approaches

b) Top-down and Bottom-up approaches

c) Symmetric and Asymmetric approaches

d) Front-end and Back-end approaches

13. Which mates define geometric relationships between assembly components? [৭]

a) Features b) Mates c) Patterns d) Sketches

14.The "Coincident" mate in Solid Works aligns two faces or edges: [c]

a) To make them parallel

b) To make them coincident

c) To make them perpendicular

d) To make them tangent

15. The "Lock" mate is used to: [a]

a) Prevent movement of the component b) Create a hinge-like connection

c) Maintain parallelism between components d) Force components to be concentric

16. The "Path Mate" in Solid Works is used to align a component with: [a]

a) A linear path b) A curved path

c) A random path d) A path in another assembly

17. The process of changing the orientation of a component around its axes [b]

Is known as:

a) Replacing b) Rotating c) Mirroring d) Moving

18. Collision Detection in Solid Works helps identify: [a]

a) Possible design issues b) Visual artifacts

c) Texture problems d) Animation errors

19. The "Coincident" mate in Solid Works aligns two entities: [C]

a) Along a common axis b) In a parallel orientation

c) With a coincident face or point d) At a specified angle

20. The purpose of the "Parallel" mate is to ensure: [b]

a) Two components are tangent to each other

b) Two components share a common axis

c) Two components are positioned at a specific distance

d) Two components have equal lengths

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DEPARTMENT OF MECHANICAL ENGINEERING
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SOLID WORKS
05/05/2022 TO 23/05/2022
ASSESSMENT TEST

12

Roll Number: 199Y5A0334 **Name of the Student:** S. Gurappa

Time: 20 Min **(Objective Questions)** **Max.Marks:** 20

Note: Answer the following Questions and each question carries **one** mark.

1. What is the purpose of the Solid Works User Interface? [c]

a) To create and edit document b) To manage files and folders

c) To browse the internet d) to play games

2. Which section of the Solid Works interface is typically used for creating and Modifying 3D models? [b]

a) Design Library

b) Feature Manager Design Tree

c) Task Pane

d) Command Manager

3. What is the purpose of a drawing template in Solid Works? [b]

a) To store sketch entities

b) To define the layout of a drawing sheet

c) To create 3D models

d) To generate G-code for CNC machines

4. What is the purpose of the Fillet tool in sketching? [a]

a) To add text annotations b) To create rounded corners

c) To draw straight lines d) To insert images

5. What is the primary function of the Solid Works User Interface? [a]

a) Gaming b) Browsing the internet

c) Creating and editing documents d) Managing emails

6. Which area of the Solid Works interface is used for managing feature history and components? [b]

a) Task Pane
b) Feature Manager Design Tree
c) Design Library
d) Command Manager

7. How can you customize Solid Works to better suit your preferences? [b]

a) By changing the default font
b) By adjusting settings in the Solid Works Options
c) By installing additional plugins
d) By updating your operating system

8. Which sketch tool is used to create a rounded internal corner between two lines Or curves? [a]

a) Fillet
b) Chamfer
c) Offset
d) Convert entities

9. The Offset tool in Solid Works sketching is primarily used to create: [c]

a) Copies of sketch entities
b) Symmetrical sketches
c) A parallel copy of a sketch entity
d) Tangent curves

10. The Intersection Curve tool in Solid Works sketching helps create a curve at The intersection of: [d]

a) Two planar faces
b) Two parallel lines
c) A line and an arc
d) A sketch and a reference plane

11. The "Jog Line" tool in Solid Works sketching is used to: [a]

a) Create zigzag patterns
b) Add symbols to the sketch

c) Generate text

d) Offset a portion of a line

12. What are the two main approaches to assembly modeling in Solid Works? [b]

a) Standard and Advanced approaches

b) Top-down and Bottom-up approaches

c) Symmetric and Asymmetric approaches

d) Front-end and Back-end approaches

13. Which mates define geometric relationships between assembly components? [a]

a) Features b) Mates c) Patterns d) Sketches

14.The "Coincident" mate in Solid Works aligns two faces or edges: [b]

a) To make them parallel

b) To make them coincident

c) To make them perpendicular

d) To make them tangent

15. The "Lock" mate is used to: [c]

a) Prevent movement of the component b) Create a hinge-like connection

c) Maintain parallelism between components d) Force components to be concentric

16. The "Path Mate" in Solid Works is used to align a component with: [a]

a) A linear path b) A curved path

c) A random path d) A path in another assembly

17. The process of changing the orientation of a component around its axes [a]

Is known as:

a) Replacing b) Rotating c) Mirroring d) Moving

18. Collision Detection in Solid Works helps identify: [c]

a) Possible design issues b) Visual artifacts

c) Texture problems d) Animation errors

19. The "Coincident" mate in Solid Works aligns two entities: [ج] ✓

a) Along a common axis b) In a parallel orientation

c) With a coincident face or point d) At a specified angle

20. The purpose of the "Parallel" mate is to ensure: [b] ✓

a) Two components are tangent to each other

b) Two components share a common axis

c) Two components are positioned at a specific distance

d) Two components have equal lengths

14

K.S.R.M. COLLEGE OF ENGINEERING (AUTONOMOUS), KADAPA-516003
DEPARTMENT OF MECHANICAL ENGINEERING
VALUE ADDED /CERTIFICATE COURSE ON
SOLID WORKS
05/05/2022 TO 23/05/2022
ASSESSMENT TEST

Roll Number: 19945A0346 **Name of the Student:** Y. Bramiah

Time: 20 Min **(Objective Questions)** **Max.Marks:** 20

Note: Answer the following Questions and each question carries **one** mark.

1. What is the purpose of the Solid Works User Interface? [c] ✓

a) To create and edit document b) To manage files and folders

c) To browse the internet d) to play games

2. Which section of the Solid Works interface is typically used for creating and Modifying 3D models? [a] ✓

a) Design Library

b) Feature Manager Design Tree

c) Task Pane

d) Command Manager

3. What is the purpose of a drawing template in Solid Works? [b] ✓

a) To store sketch entities

b) To define the layout of a drawing sheet

c) To create 3D models

d) To generate G-code for CNC machines

4. What is the purpose of the Fillet tool in sketching? [a] ✓

a) To add text annotations b) To create rounded corners

c) To draw straight lines d) To insert images

5. What is the primary function of the Solid Works User Interface? [c] ✓

a) Gaming b) Browsing the internet

c) Creating and editing documents d) Managing emails

6. Which area of the Solid Works interface is used for managing feature history and components? [b]

a) Task Pane
b) Feature Manager Design Tree
c) Design Library
d) Command Manager

7. How can you customize Solid Works to better suit your preferences? [b]

a) By changing the default font
b) By adjusting settings in the Solid Works Options
c) By installing additional plugins
d) By updating your operating system

8. Which sketch tool is used to create a rounded internal corner between two lines Or curves? [c]

a) Fillet
b) Chamfer
c) Offset
d) Convert entities

9. The Offset tool in Solid Works sketching is primarily used to create: [a]

a) Copies of sketch entities
b) Symmetrical sketches
c) A parallel copy of a sketch entity
d) Tangent curves

10. The Intersection Curve tool in Solid Works sketching helps create a curve at The intersection of: [a]

a) Two planar faces
b) Two parallel lines
c) A line and an arc
d) A sketch and a reference plane

11. The "Jog Line" tool in Solid Works sketching is used to: [d]

a) Create zigzag patterns
b) Add symbols to the sketch

c) Generate text

d) Offset a portion of a line

12. What are the two main approaches to assembly modeling in Solid Works? [b]

a) Standard and Advanced approaches

b) Top-down and Bottom-up approaches

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a) Two components are tangent to each other

b) Two components share a common axis

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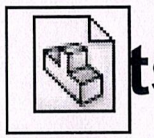
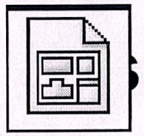
d) Two components have equal lengths

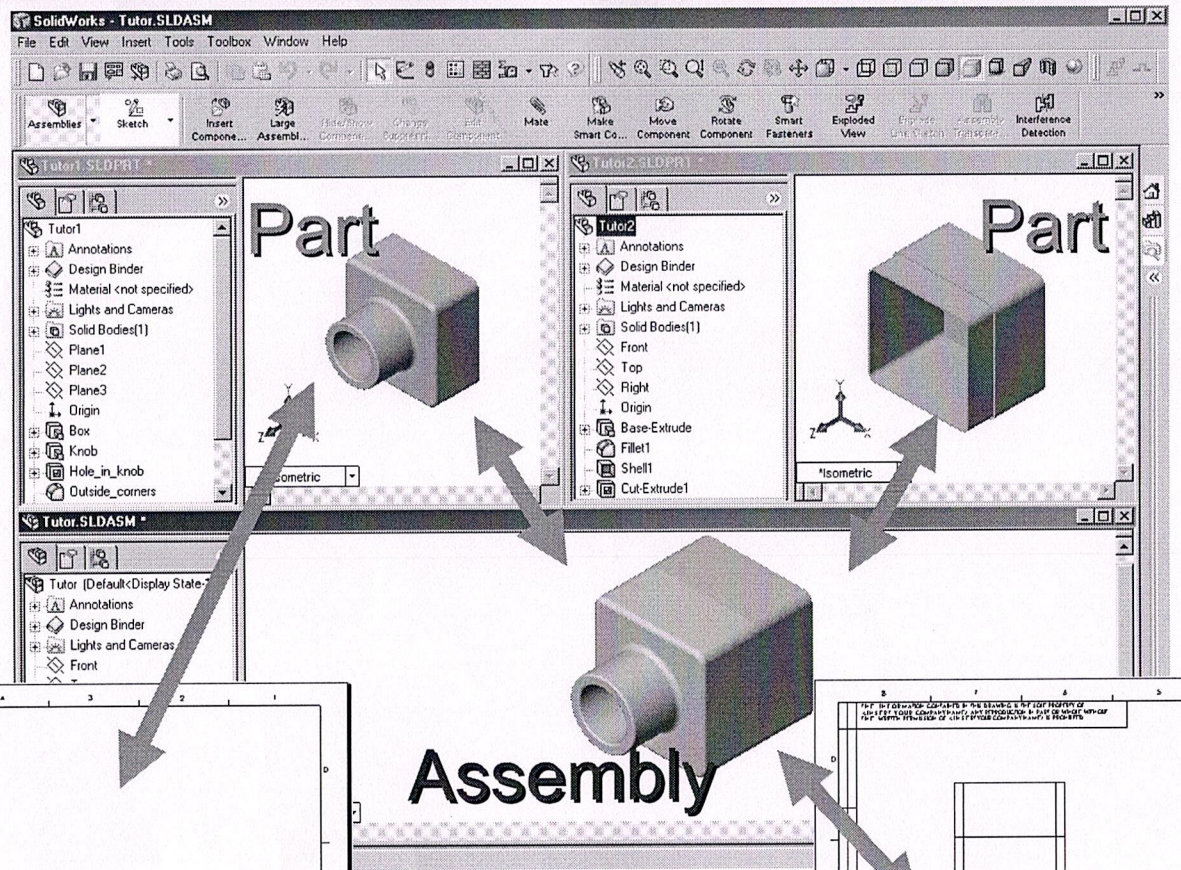
What is SolidWorks?

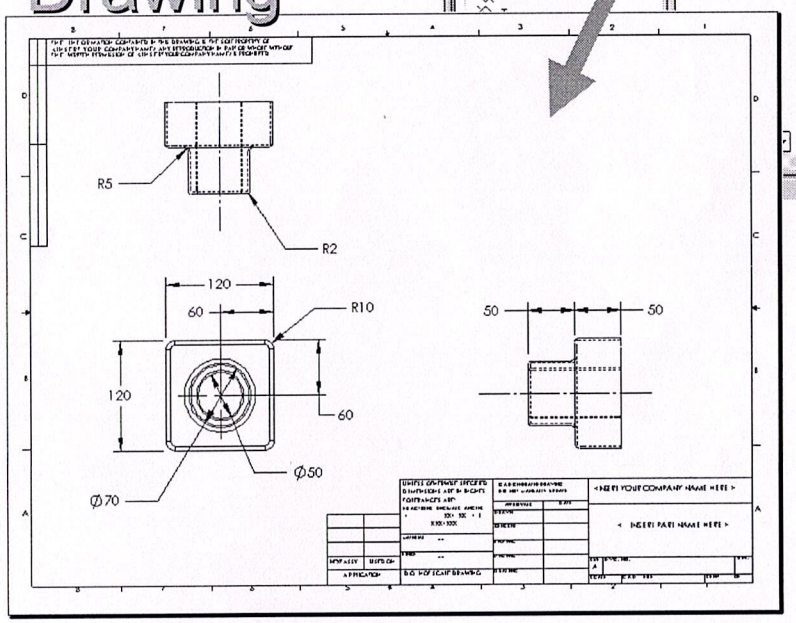
- SolidWorks is design automation software.
- In SolidWorks, you sketch ideas and experiment with different designs to create 3D models.
- SolidWorks is used by students, designers, engineers, and other professionals to produce simple and complex parts, assemblies, and drawings.

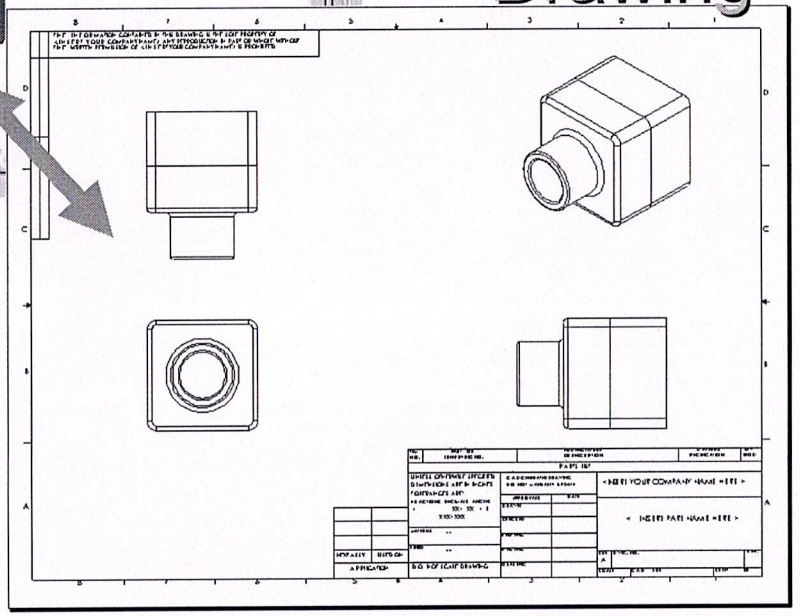
The SolidWorks Model

- **The SolidWorks model is made up of:**
  - **Parts** 
  - **Assemblies** 
  - **Drawings** 

The SolidWorks Model

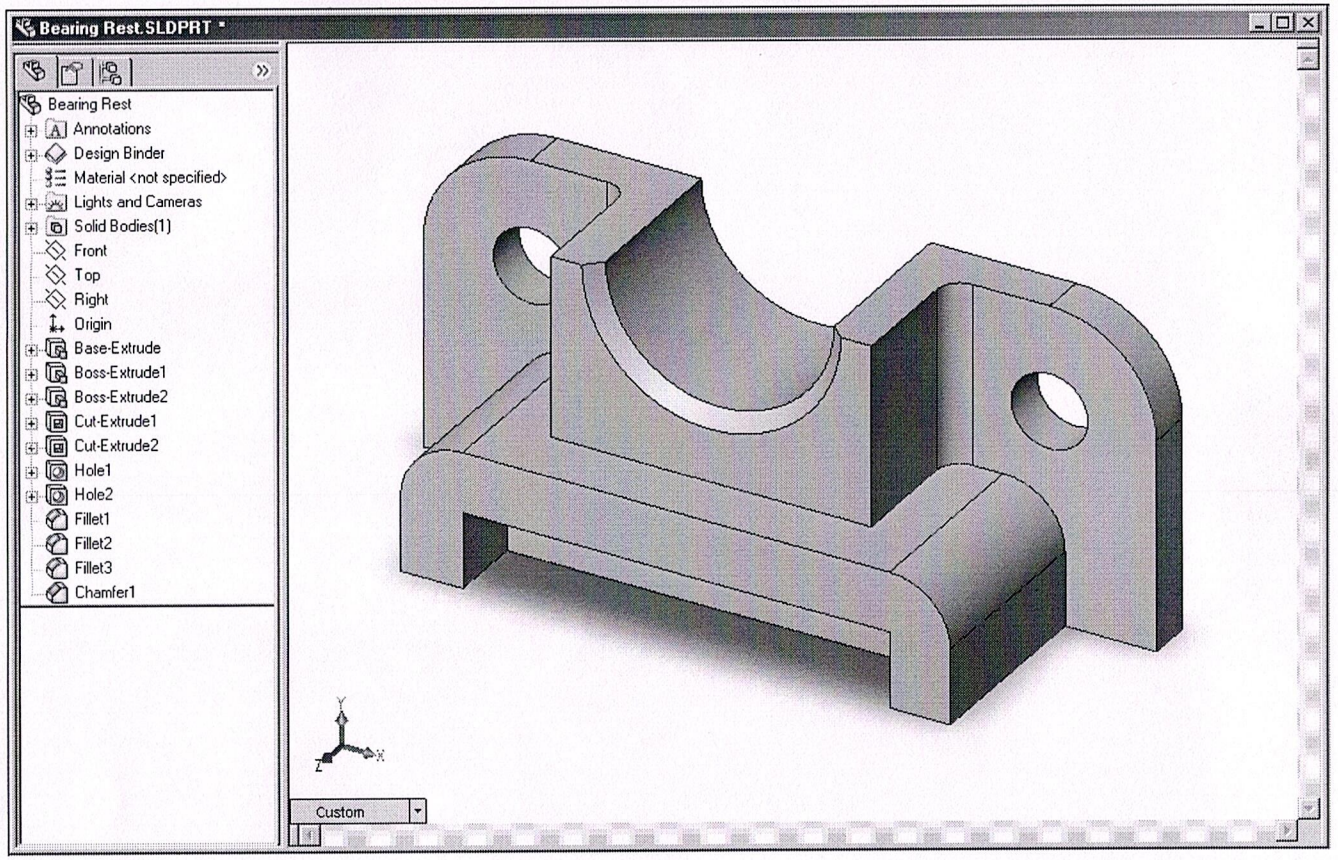






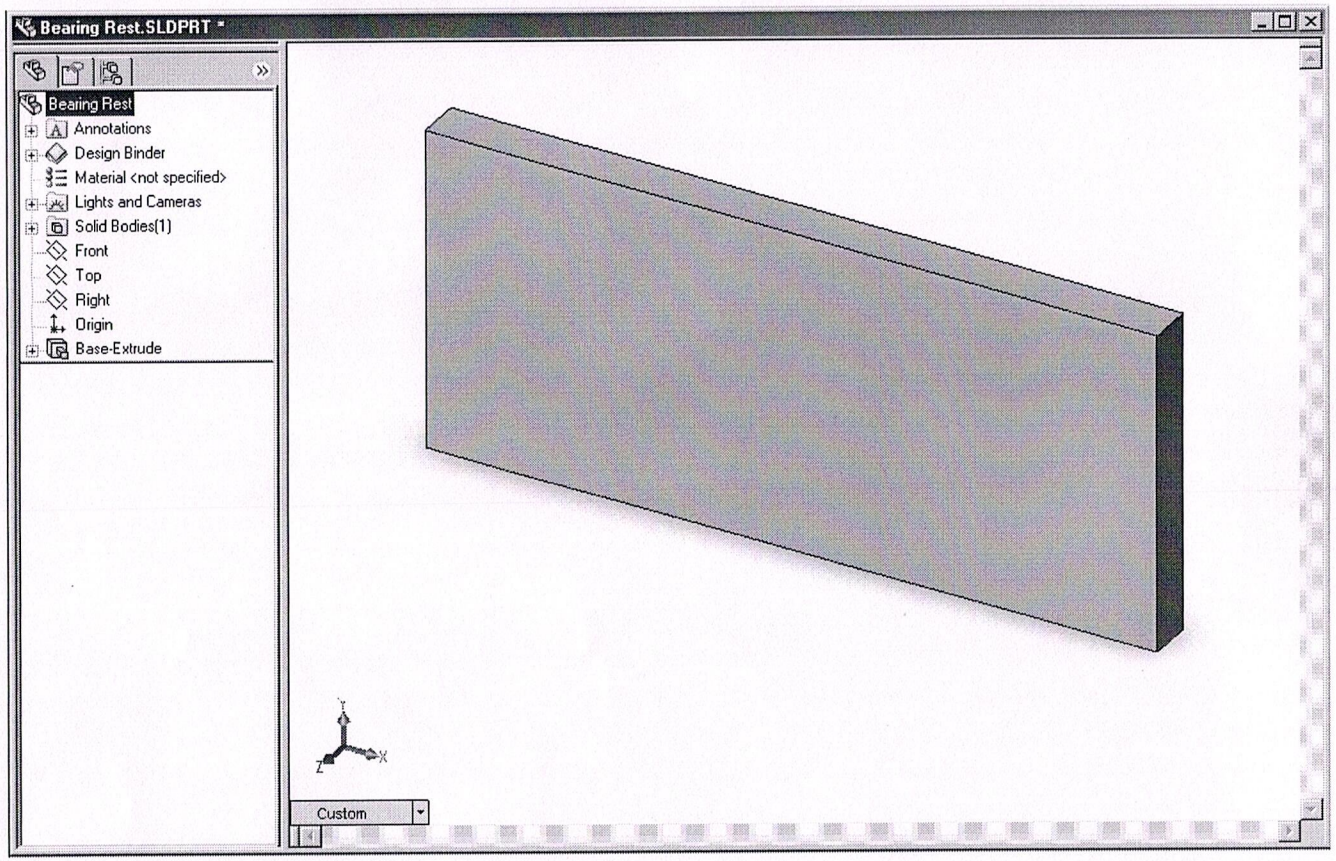
Features

- **Features are the building blocks of the part.**
- **Features are the *shapes* and *operations* that construct the part.**



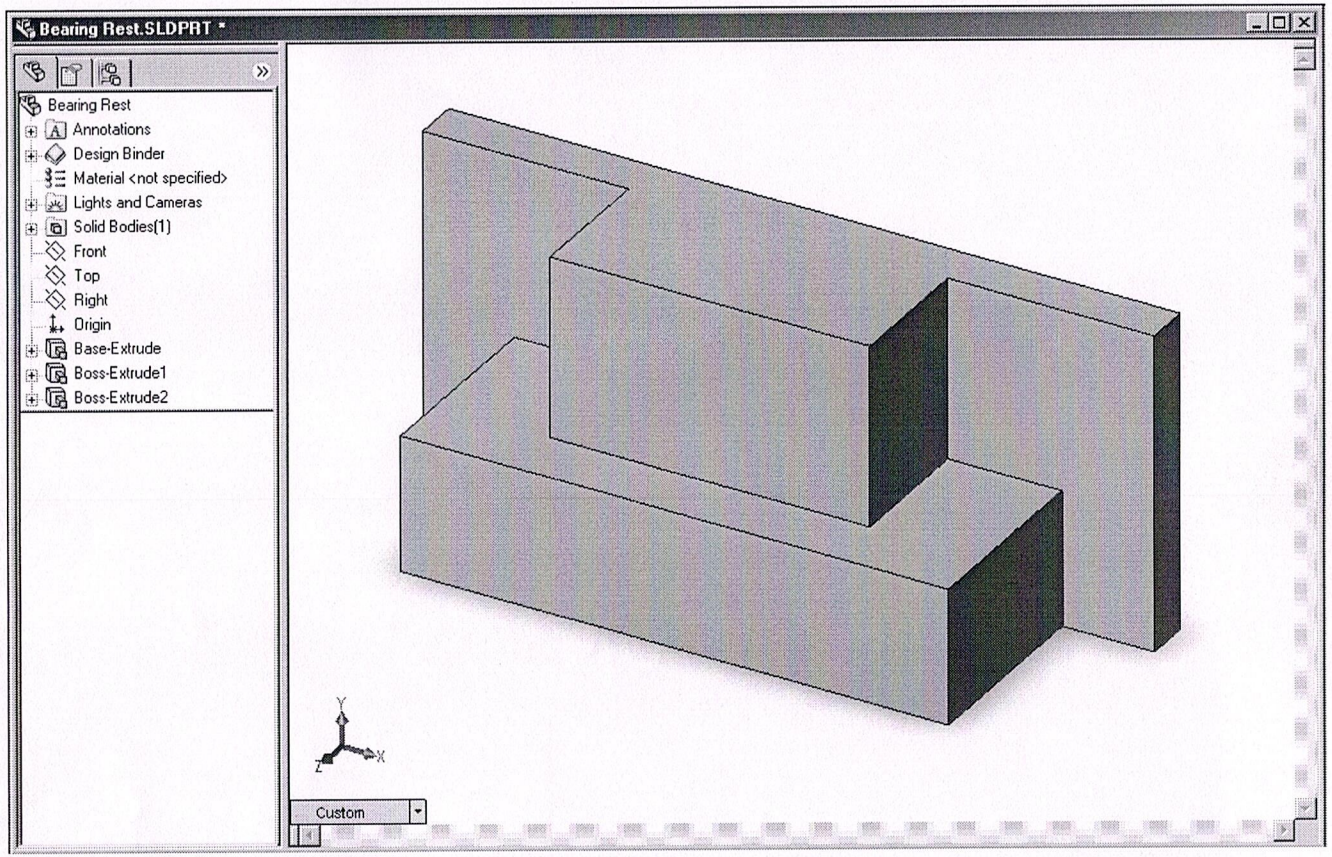
Examples of Shape Features

- **Base Feature**
  - First feature in part.
  - Created from a 2D sketch.
  - Forms the work piece to which other features are added.



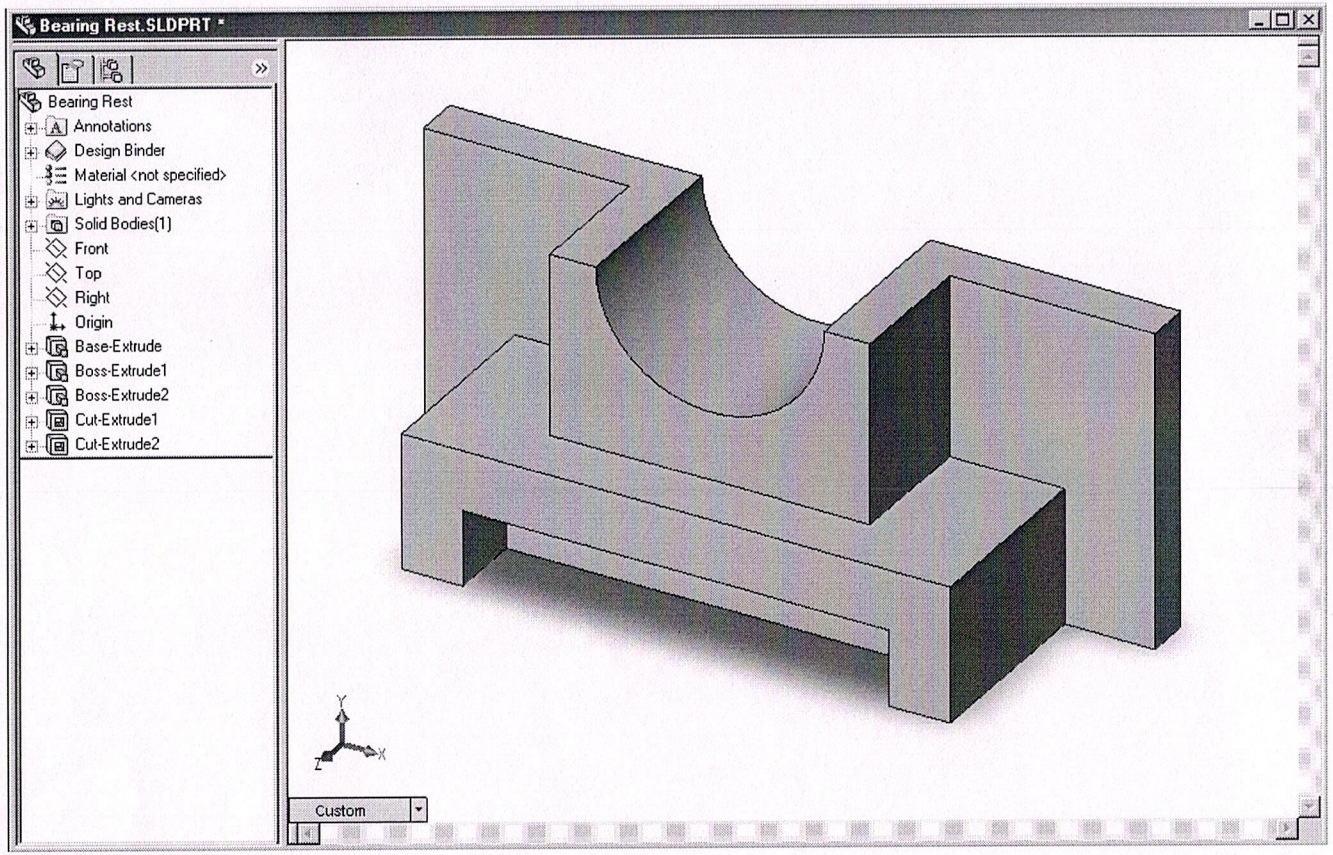
Examples of Shape Features

- **Boss feature**
  - **Adds material to part.**
  - **Created from 2D sketch.**



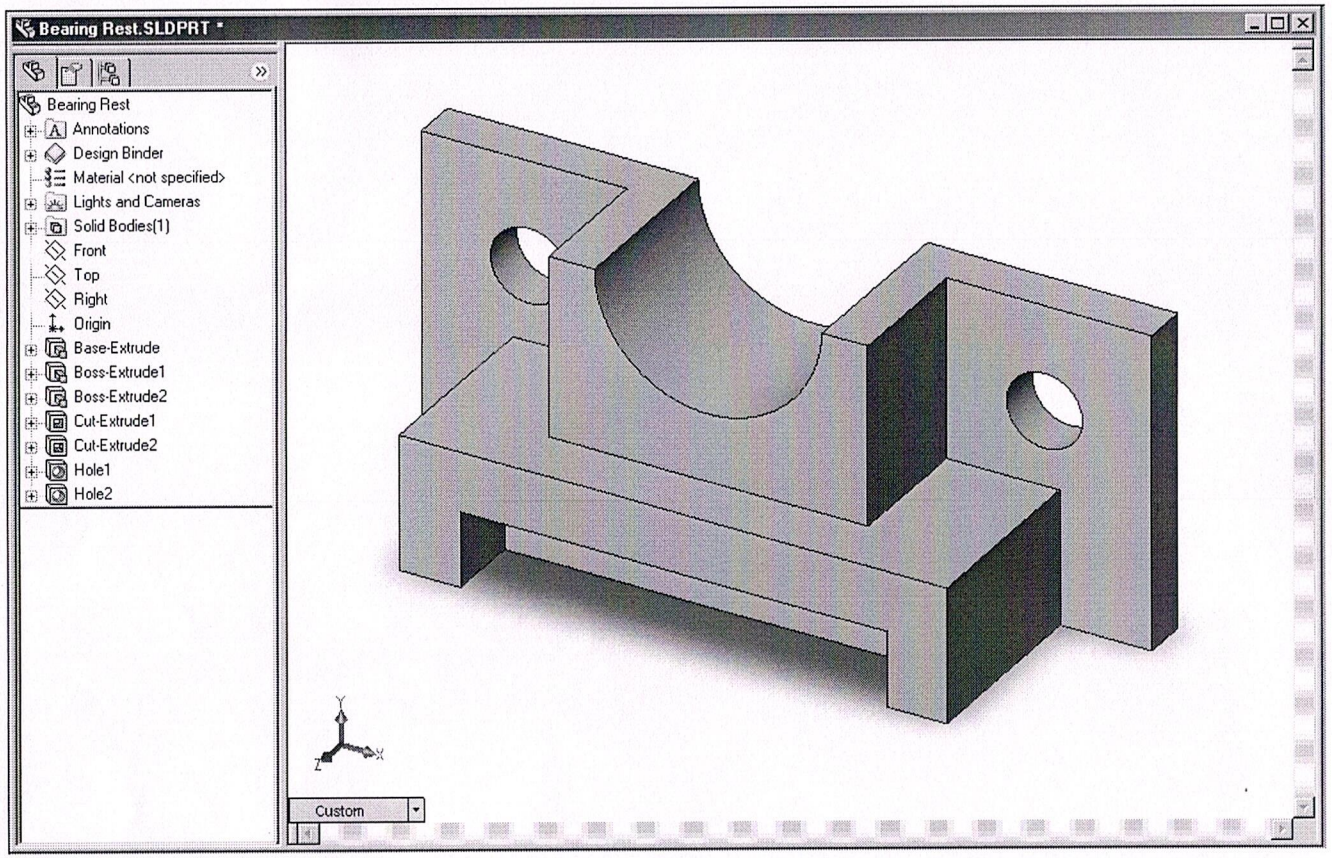
Examples of Shape Features

- **Cut feature**
  - **Removes material from part.**
  - **Created from 2D sketch.**



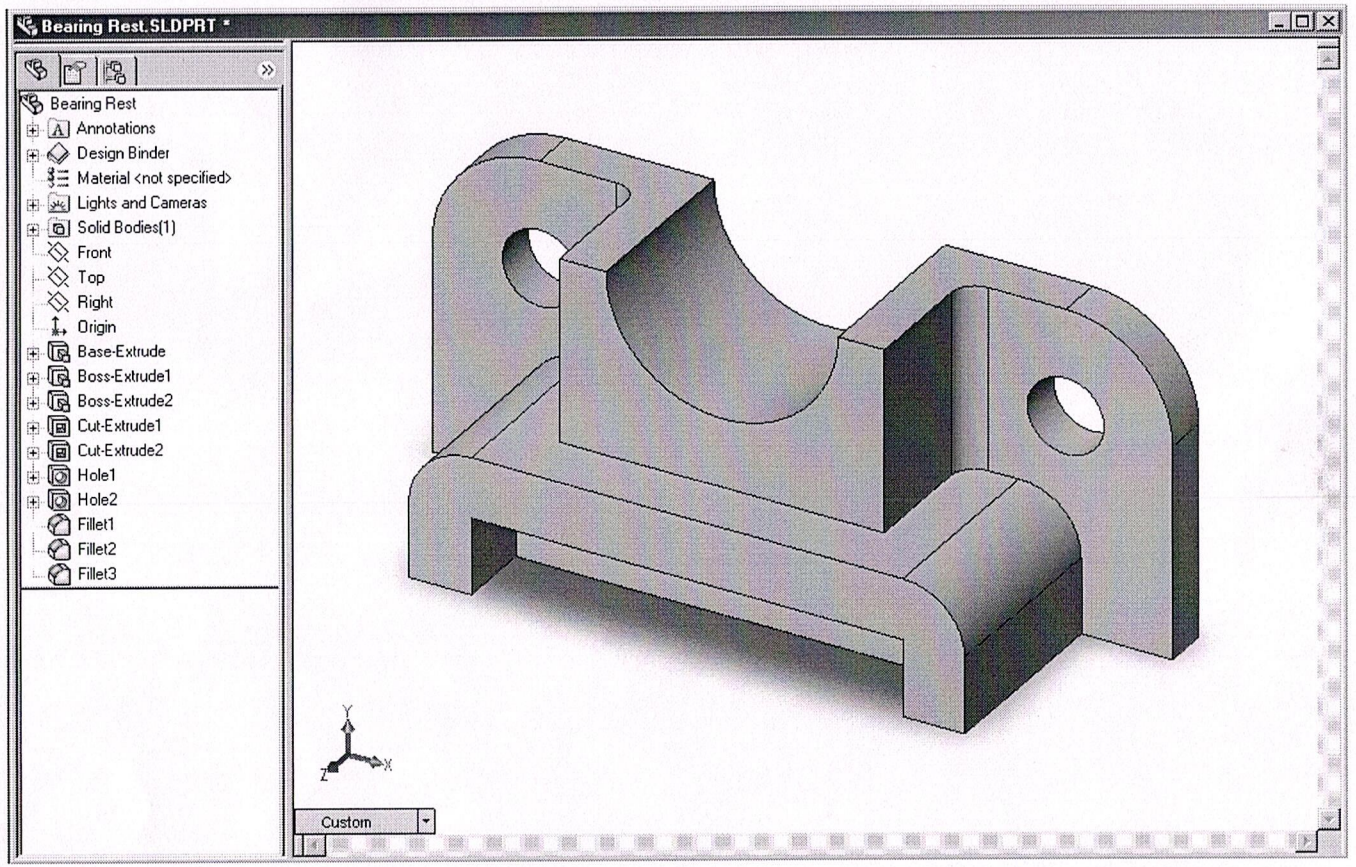
Examples of Shape Features

- **Hole feature**
  - **Removes material.**
  - **Works like more intelligent cut feature.**
  - **Corresponds to process such as counter-sink, thread, counter-bore.**



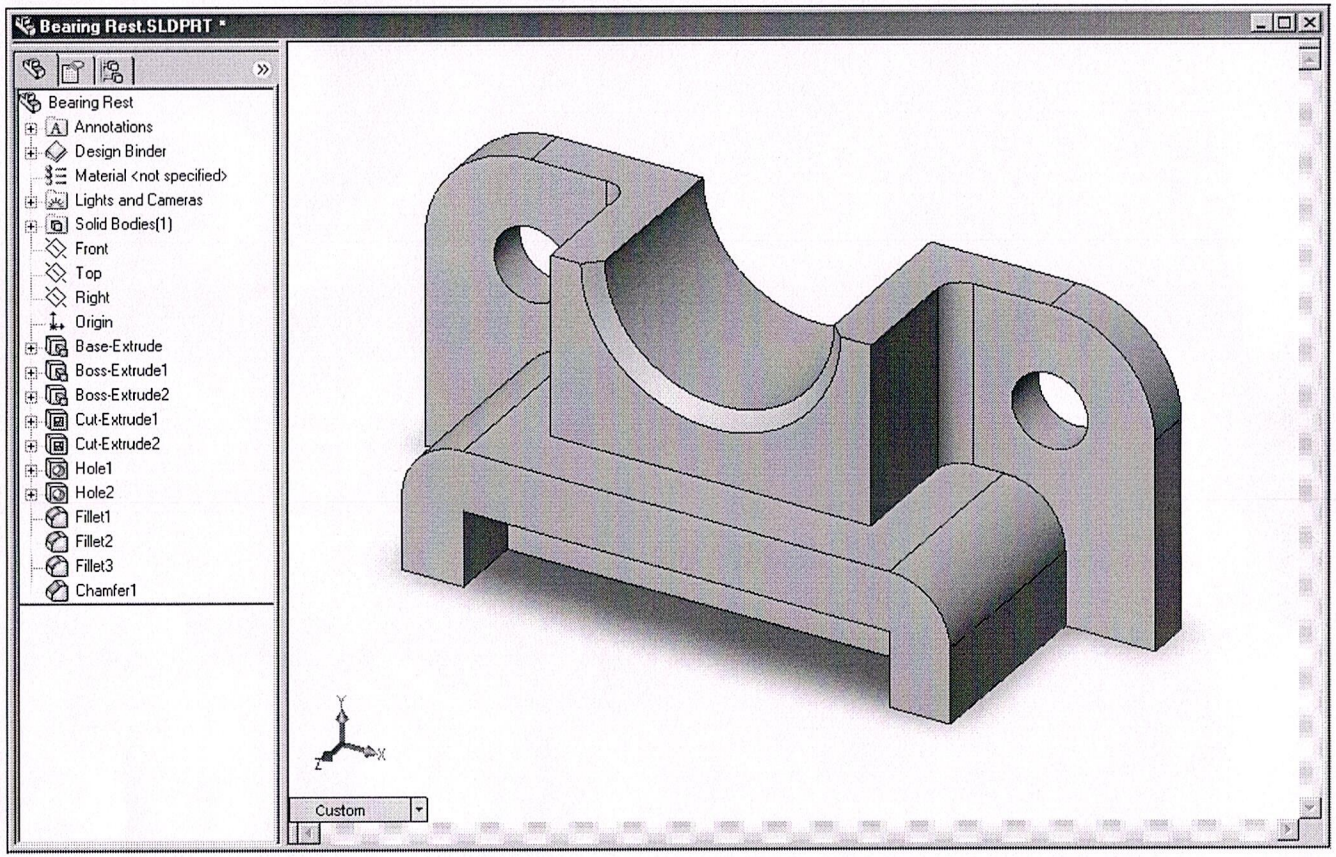
Examples of Shape Features

- **Fillet feature**
  - **Used to round off sharp edges.**
  - **Can remove or add material.**
    - **Outside edge (convex fillet) removes material.**
    - **Inside edge (concave fillet) adds material.**



Examples of Shape Features

- **Chamfer feature**
  - **Similar to a fillet.**
  - **Bevels an edge rather than rounding it.**
  - **Can remove or add material.**



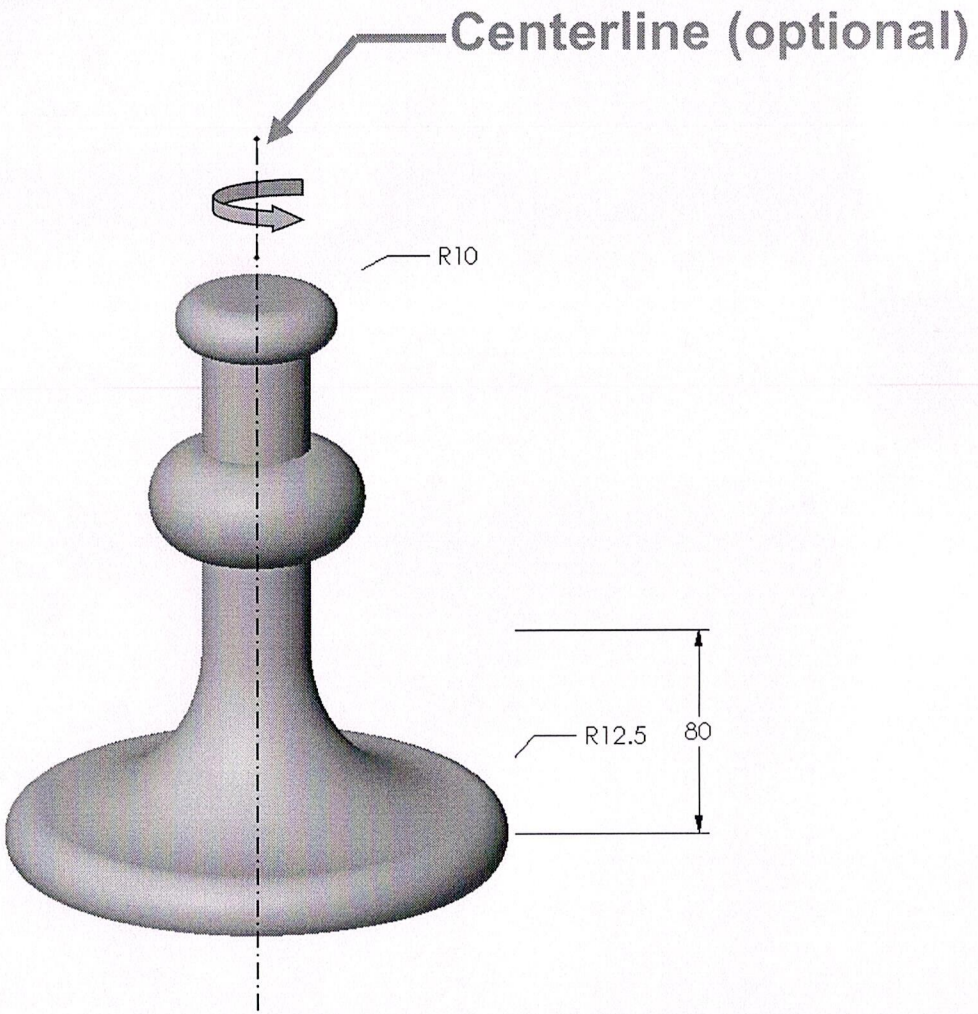
Sketched Features & Operation Features

- **Sketched Features**
  - Shape features have sketches.
  - Sketched features are built from 2D profiles.

- **Operation Features**
  - Operation features do not have sketches.
  - Applied directly to the work piece by selecting edges or faces.

To Create a Revolved Base Feature:

1. **Select a sketch plane.**
2. **Sketch a 2D profile.**
3. **Sketch a centerline (optional).**
4. **Revolve the sketch around a sketch line or centerline.**

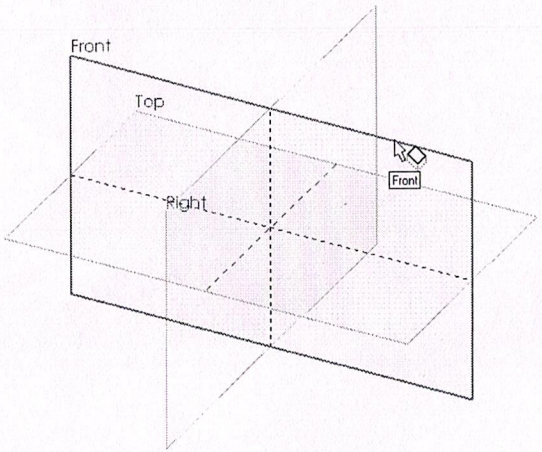


To Create an Extruded Base Feature:

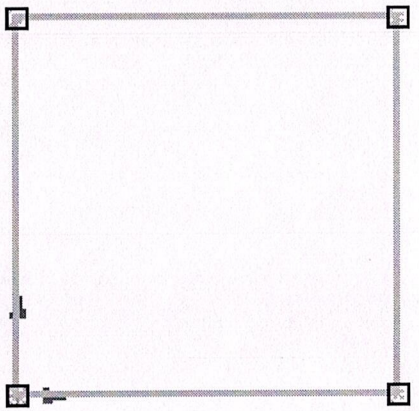
1. **Select a sketch plane.**

2. **Sketch a 2D profile.**

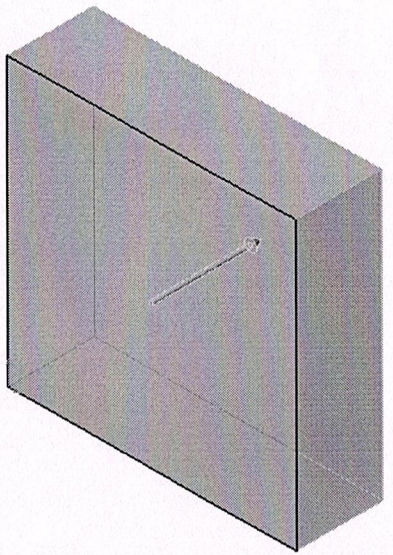
3. **Extrude the sketch perpendicular to sketch plane.**



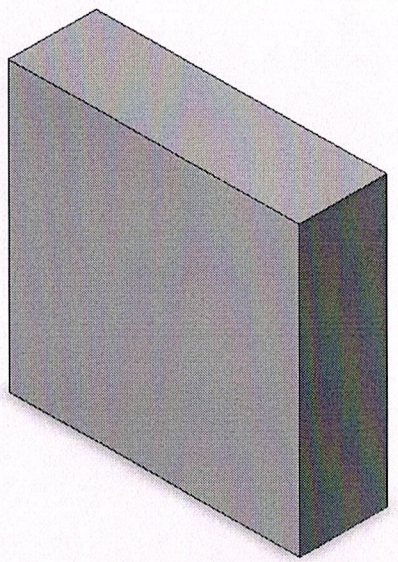
Select the sketch plane



Sketch the 2D profile



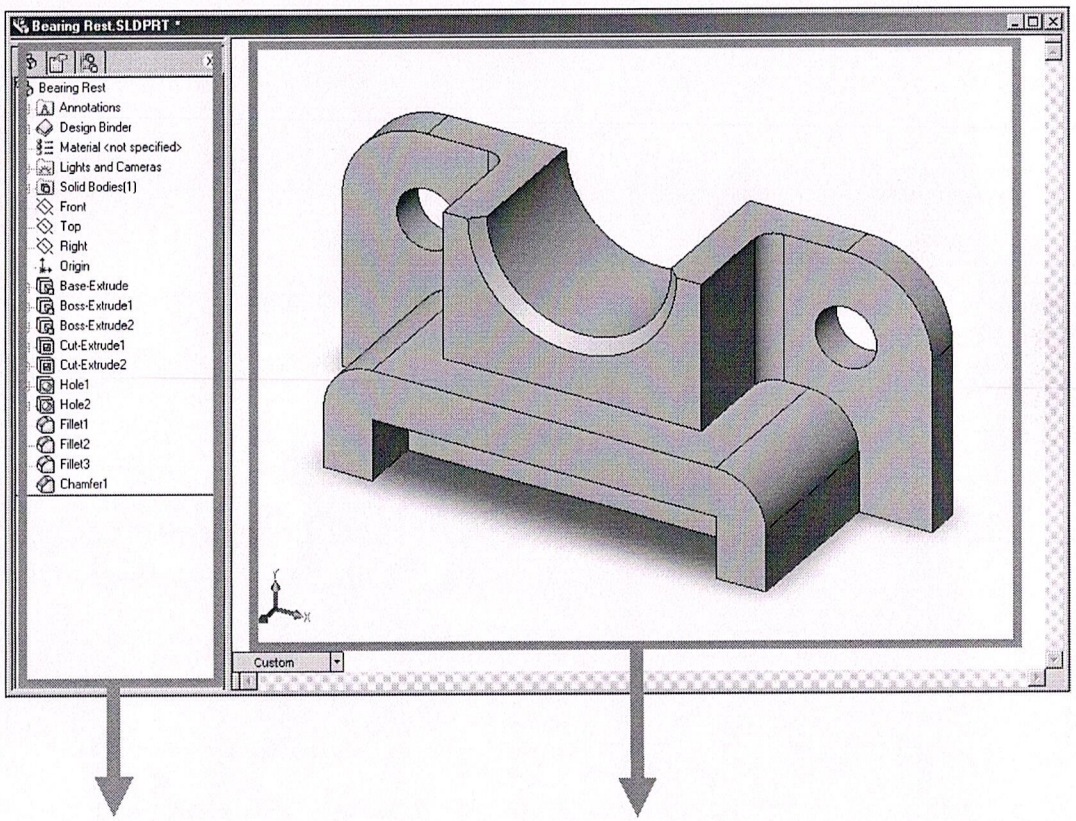
Extrude the sketch



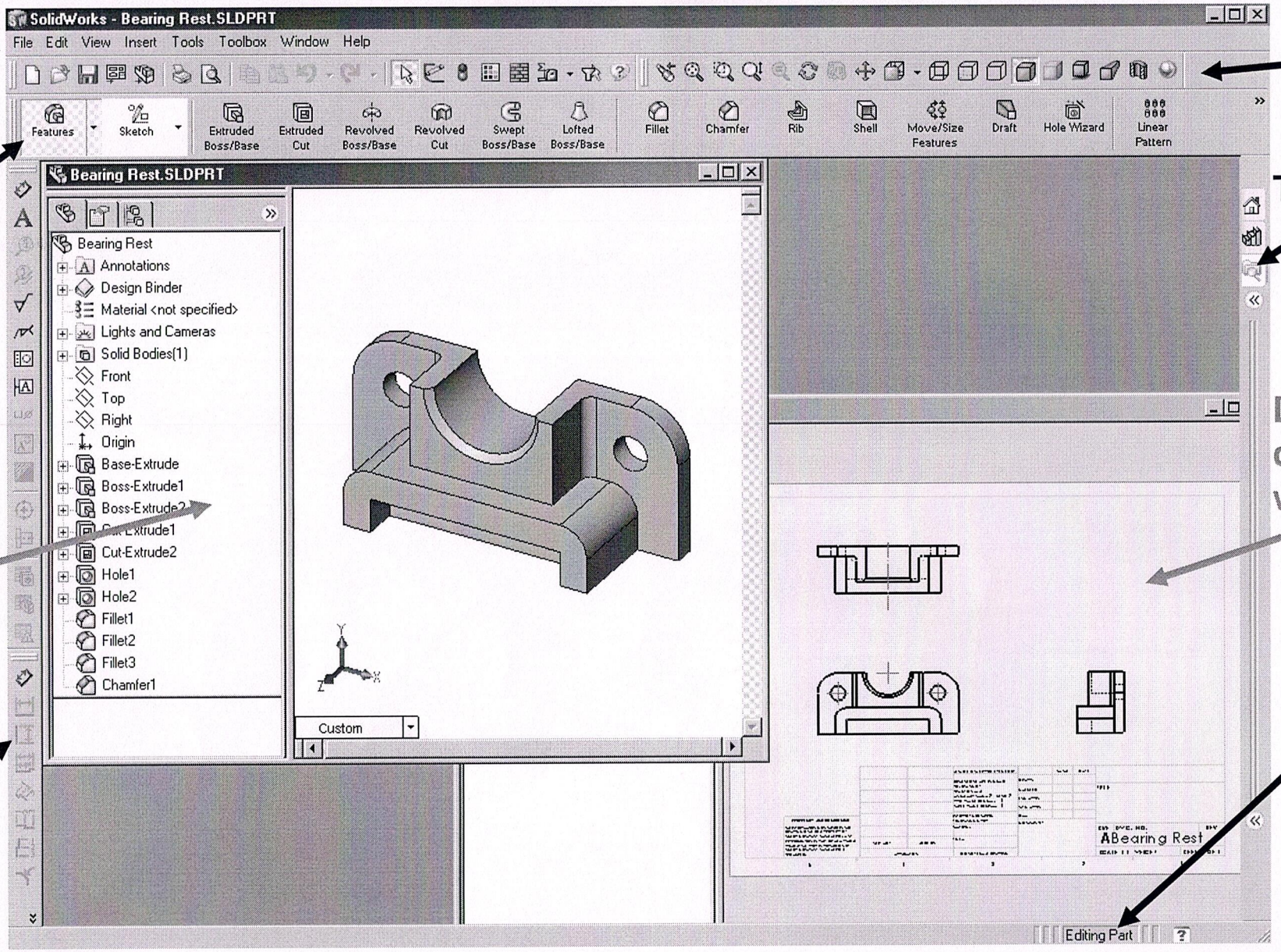
Resulting base feature

Terminology: Document Window

- **Divided into two panels:**
  - **Left panel contains the FeatureManager® design tree.**
    - **Lists the structure of the part, assembly or drawing.**
  - **Right panel contains the Graphics**
    - **Location to display, create, and modify a part, assembly or drawing.**



Terminology: User Interface



Menu Bar

Toolbar

Command Manager

Task pane

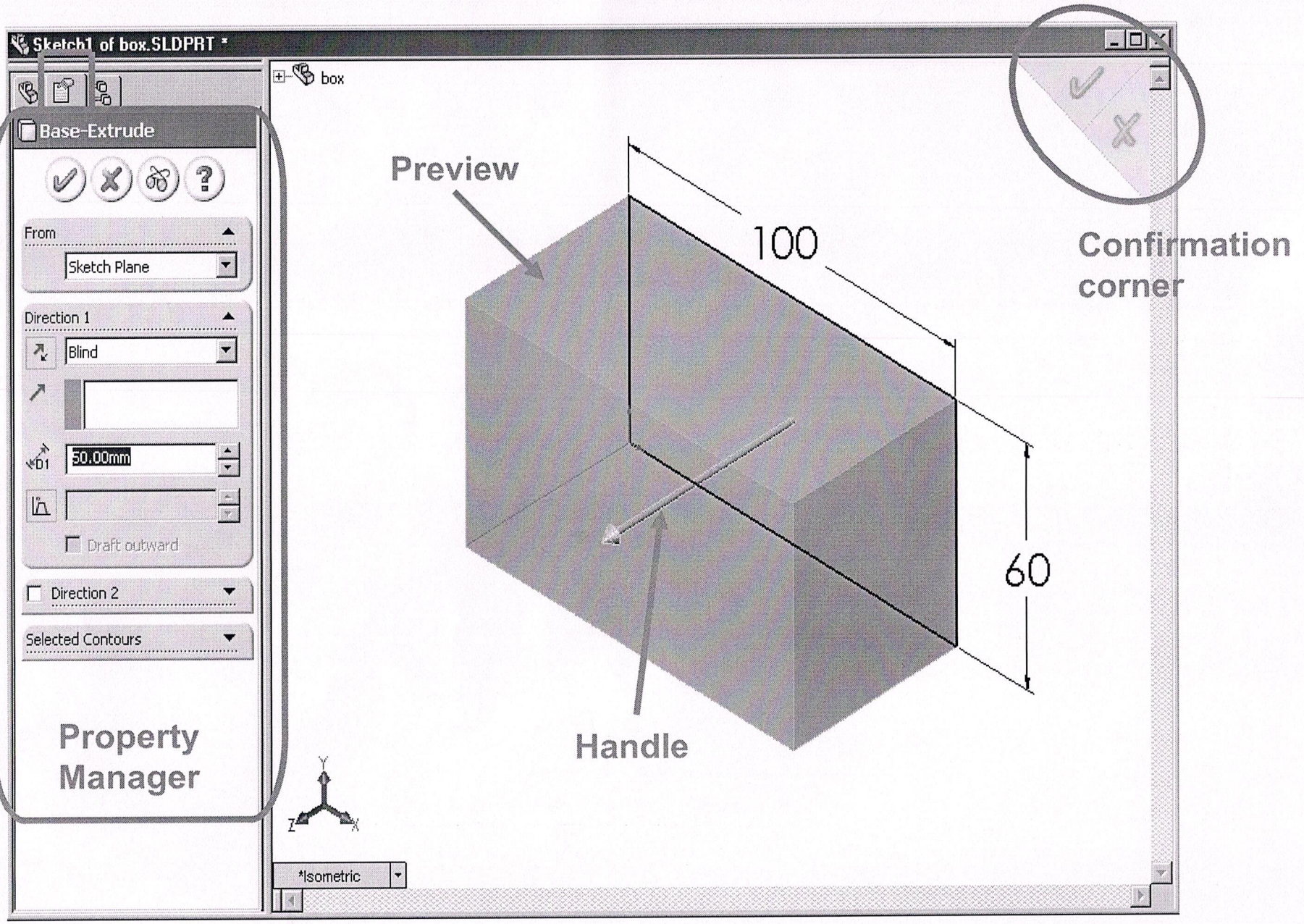
Drawing document window

Part document window

Toolbar

Status bar

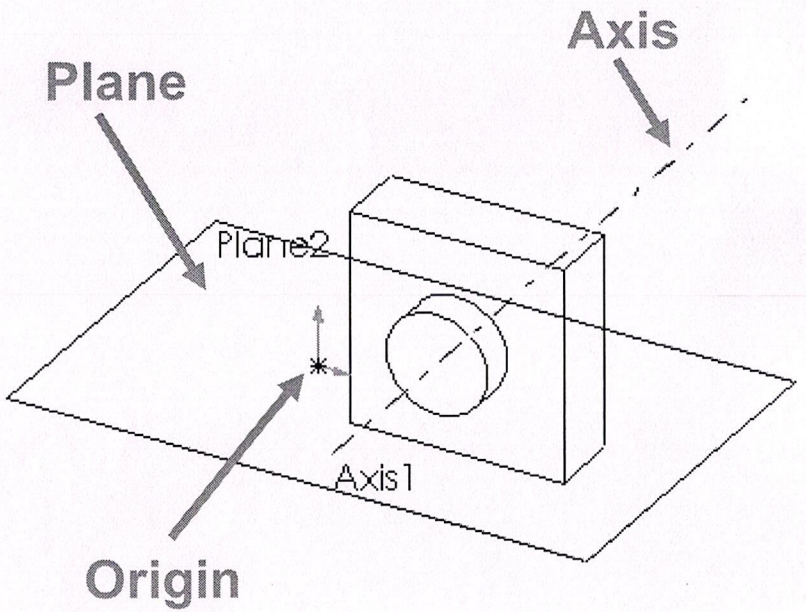
Terminology: PropertyManager



Terminology: Basic Geometry

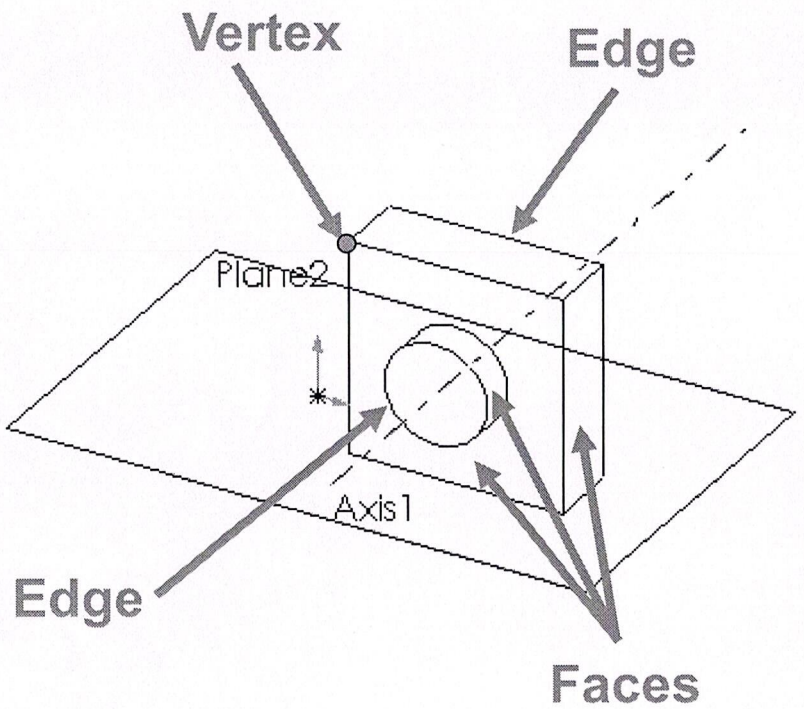
- **Axis - An implied centerline that runs through every cylindrical feature.**
- **Plane - A flat 2D**
- **Origin - The point where the three default reference planes intersect. The coordinates of the origin are:**

  (x = 0, y = 0, z = 0).



Terminology: Basic Geometry

- **Face**
  The surface or "skin" of a part. Faces can be flat or curved.
- **Edge**
  The boundary of a face. Edges can be straight or curved.
- **Vertex**
  The corner where edges meet.



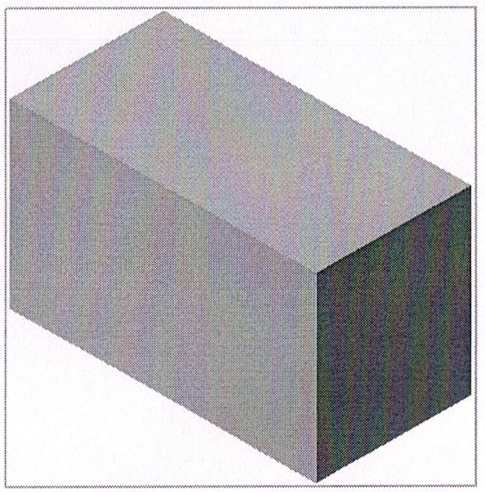
Base feature

- **The Base feature is the first feature that is created.**
- **The Base feature is the foundation of the part.**
- **The Base feature geometry for the box is an extrusion.**
- **The extrusion is named** Extrude1**.**

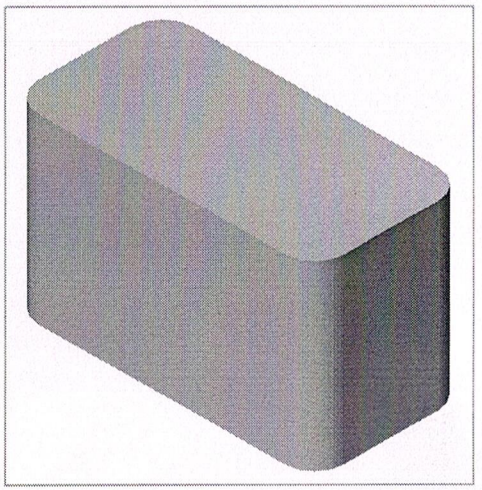
Features and Commands

Features used to build the *box* are:

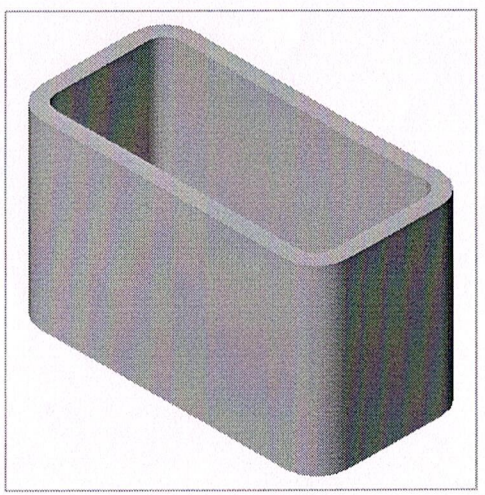
- **Extruded Base feature**
- **Fillet feature**
- **Shell feature**
- **Extruded Cut feature**



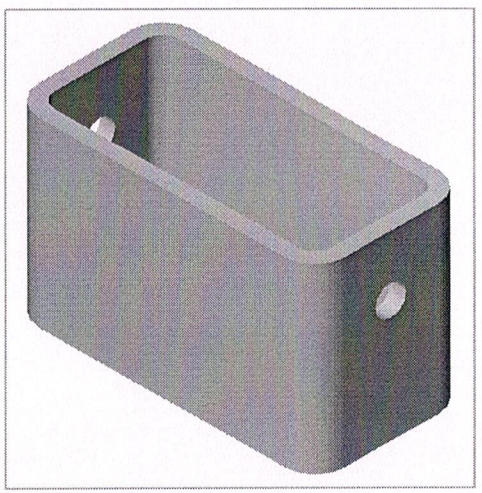
1.Base Feature



2.Fillet Feature



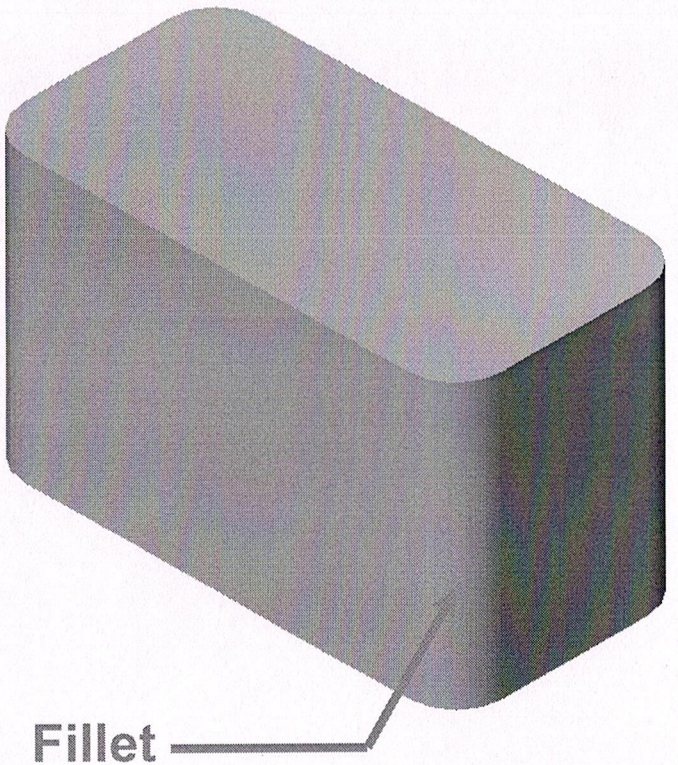
3.Shell Feature



4.Cut Feature

Fillet feature

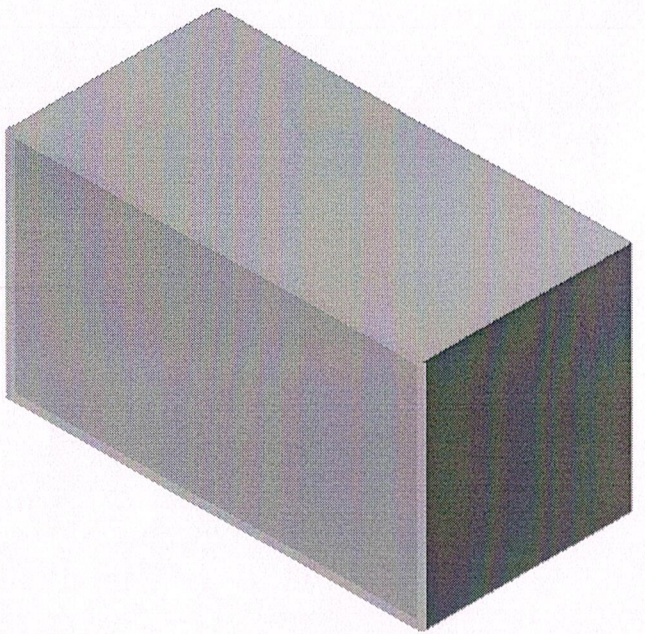
- The fillet feature rounds the edges or faces of a part.
- Select the edges to be rounded. Selecting a face rounds all the edges of that face.
- Specify the fillet radius.



Features and Commands

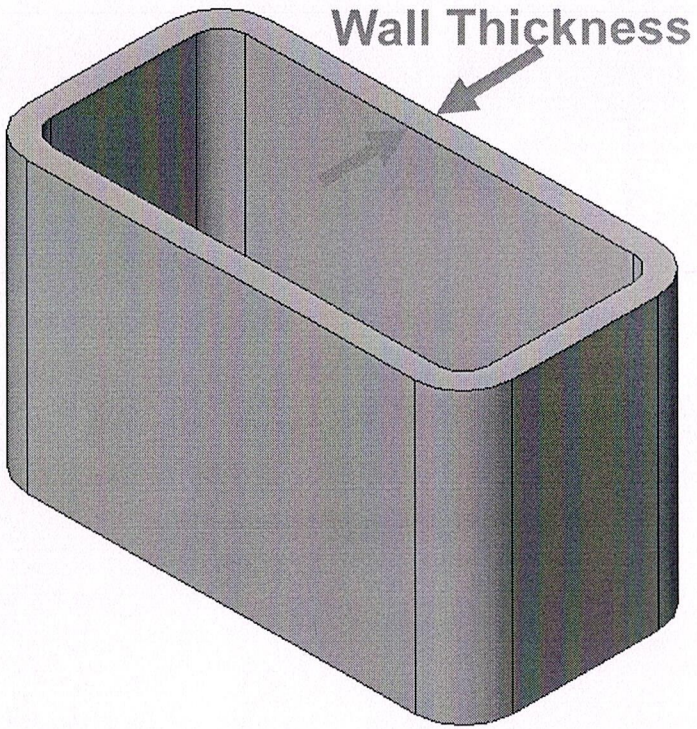
To create the extruded base feature for the *box*:

- **Sketch a rectangular profile c a 2D plane.**
- **Extrude the sketch.**
- **By default extrusions are perpendicular to the sketch plane.**



Shell feature

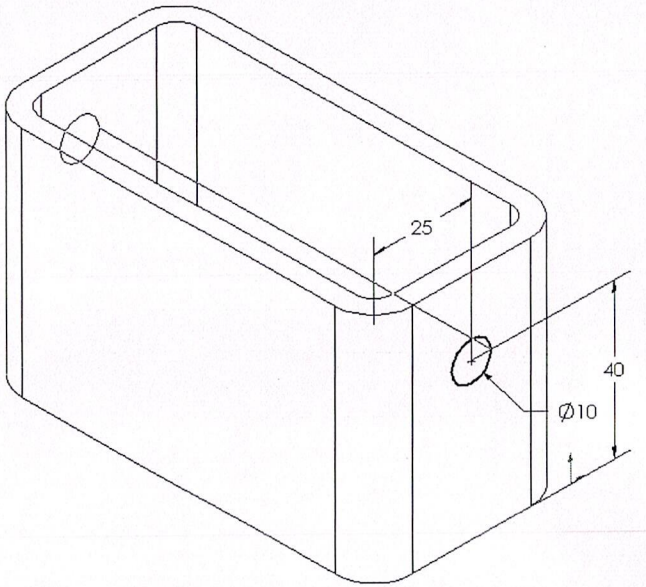
- The shell feature removes material from the selected face.
- Using the shell feature creates a hollow box from a solid box.
- Specify the wall thickness for the shell feature.

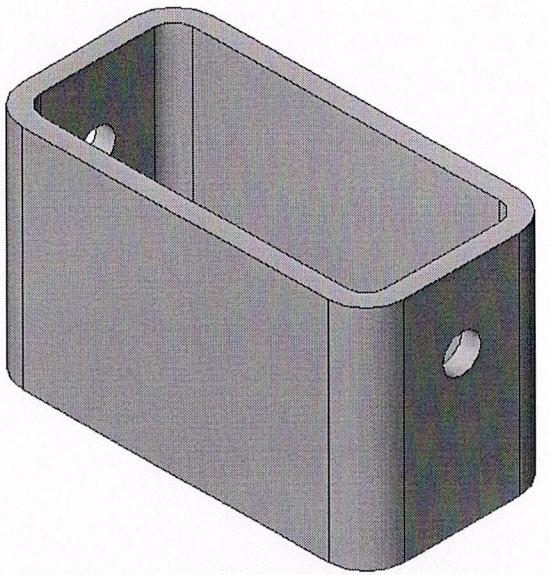


Features and Commands

To create the extruded cut feature for the *box*:

- **Sketch the 2D circular profile.**
- **Extrude the 2D Sketch profile perpendicular to the sketch plane.**
- **Enter Through All for the end condition.**
- **The cut penetrates through the entire part.**



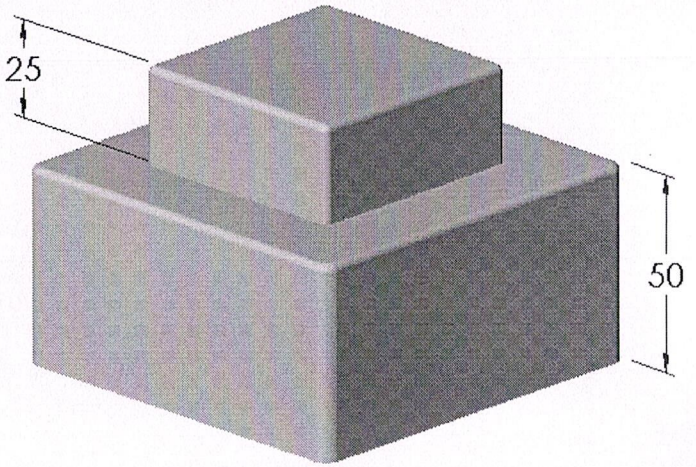


Dimensions and Geometric Relationships

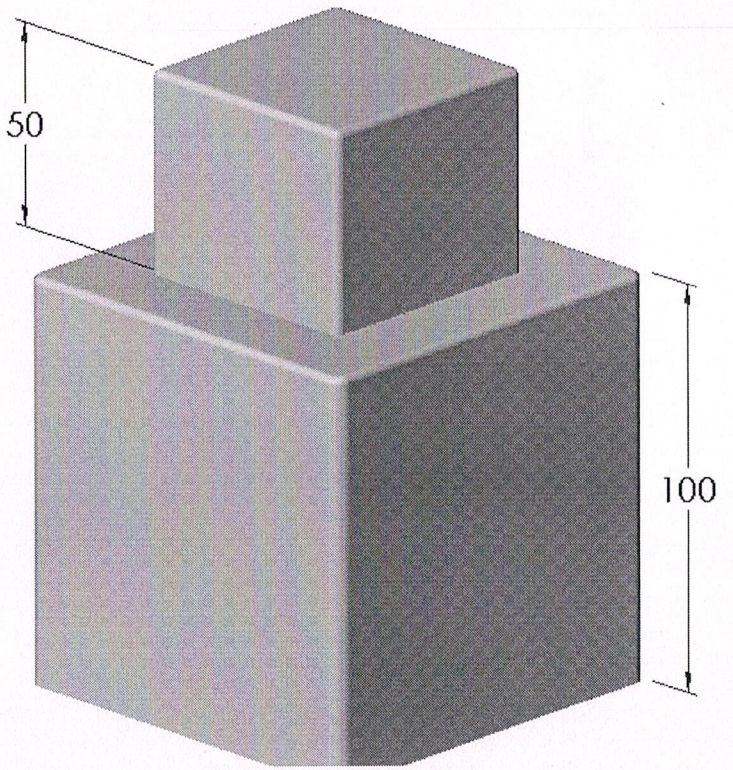
- Specify dimensions and geometric relationships between features and sketches.
- Dimensions change the size and shape of the part.
- Mathematical relationships between dimensions can be controlled by equations.
- Geometric relationships are the rules that control the behavior of sketch geometry.
- Geometric relationships help capture design intent.

Dimensions

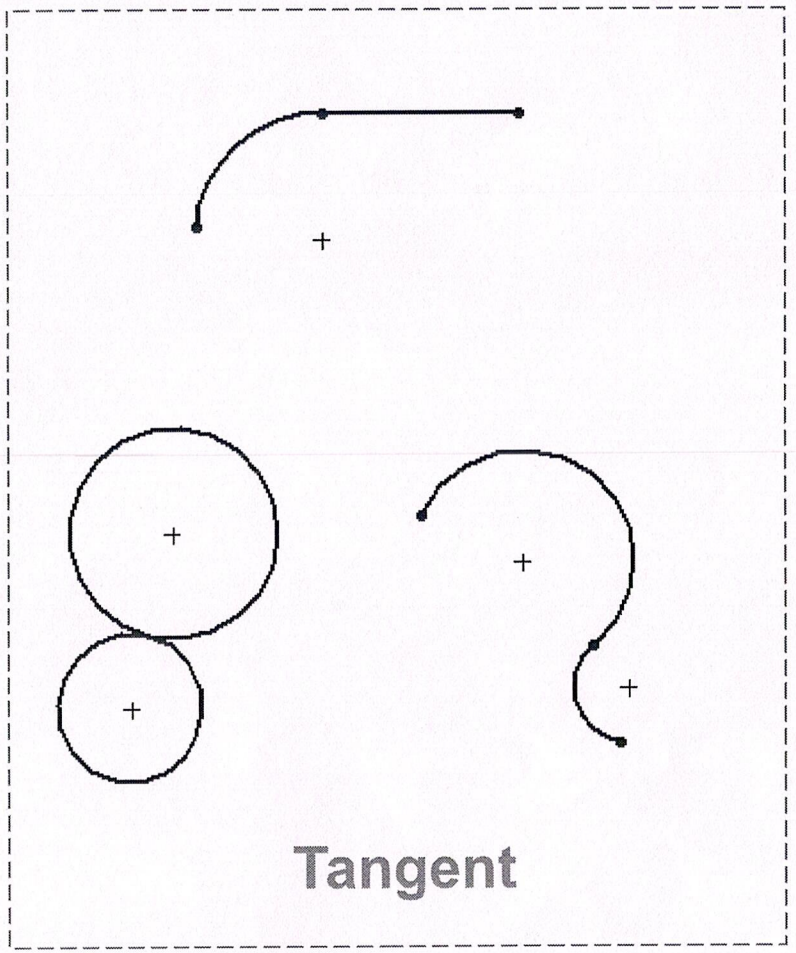
- **Dimensions**
  - **Base depth = 50 mm**
  - **Boss depth = 25 mm**



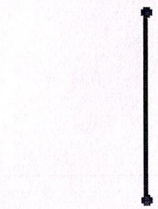
- **Mathematical relationship**
  - **Boss depth = Base depth ÷2**



Geometric Relationships

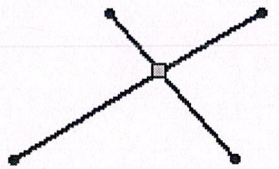


Tangent

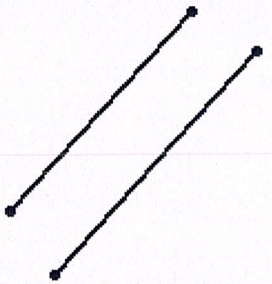


Vertical

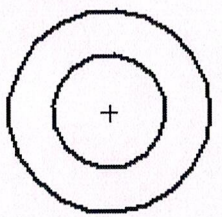
Horizontal



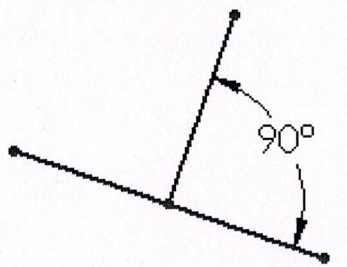
Intersection



Parallel



Concentric



Perpendicular

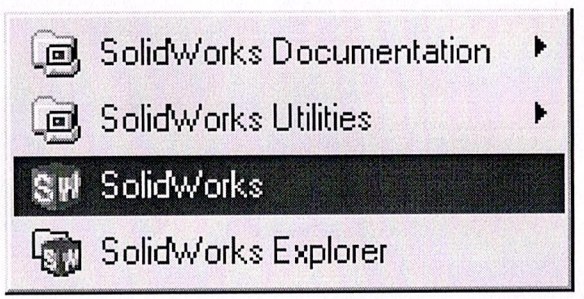
To Start SolidWorks

- **Click the Start button (Start) Windows task bar.**

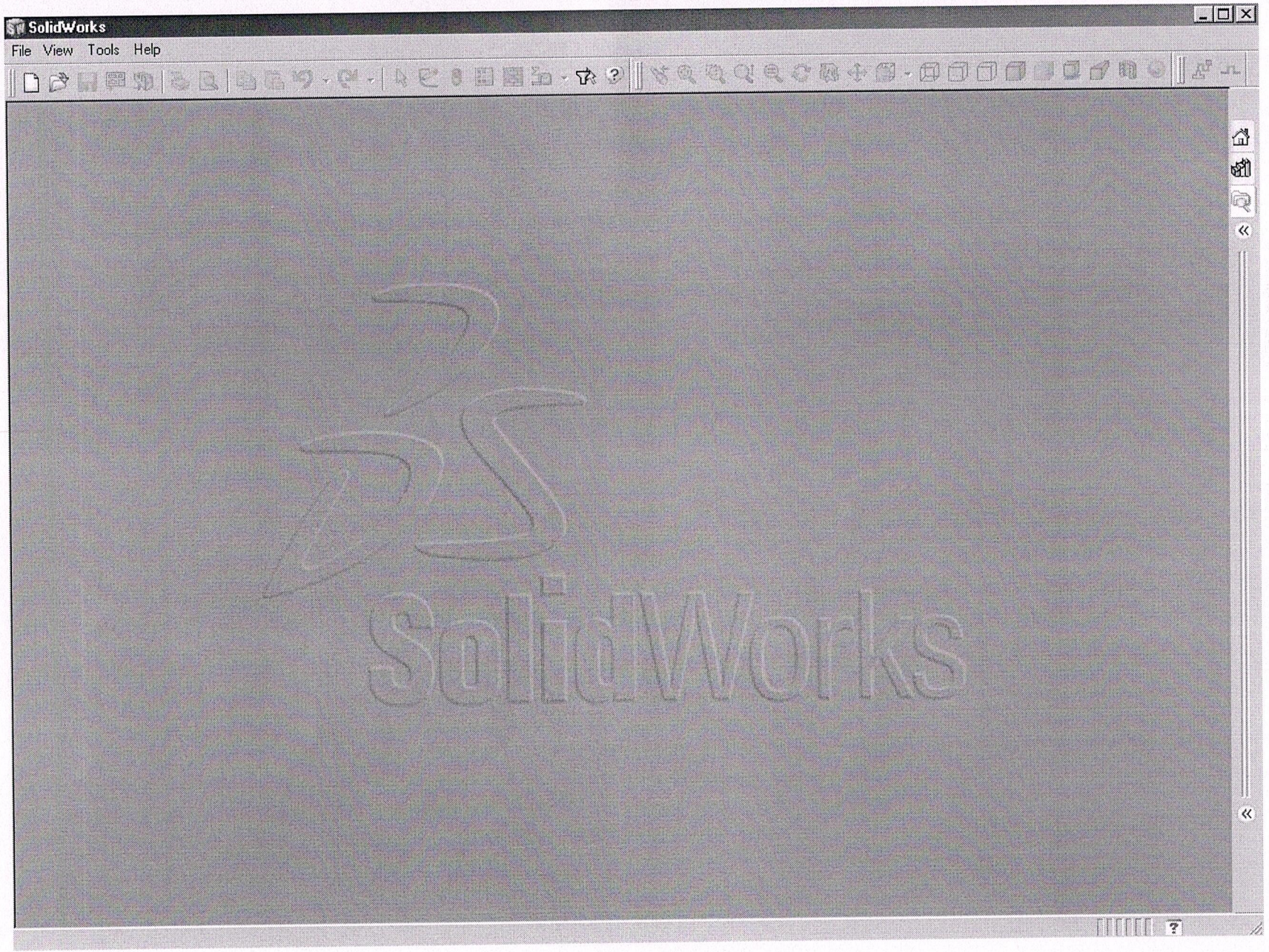
  Click Programs.

    Click the SolidWorks folder.

      Click the SolidWorks application.

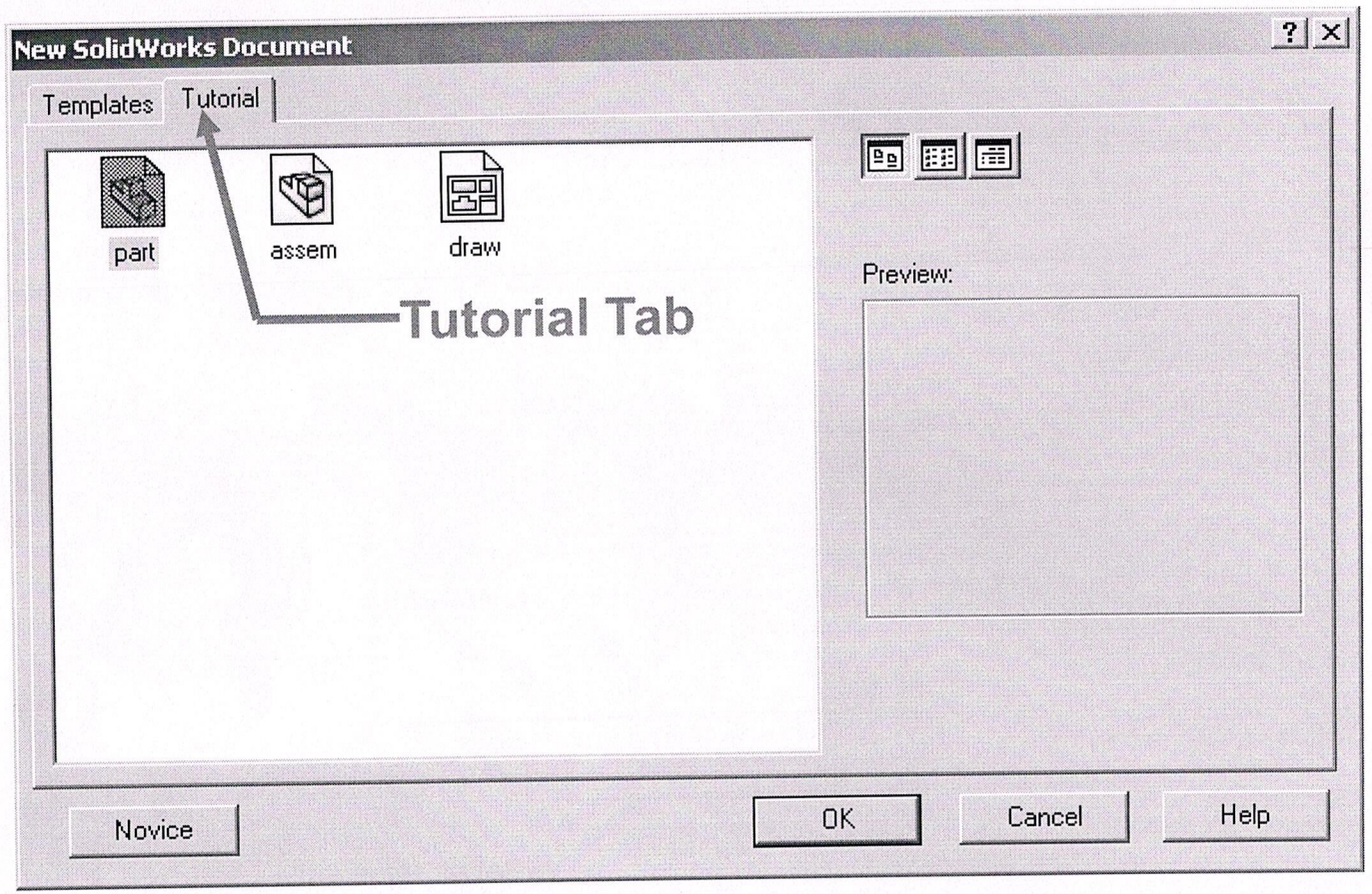


The SolidWorks Window



Creating New Files Using Templates

- **Click New the Standard toolbar.**
- **Select a document template:**
  - Part
  - Assembly
  - Drawing

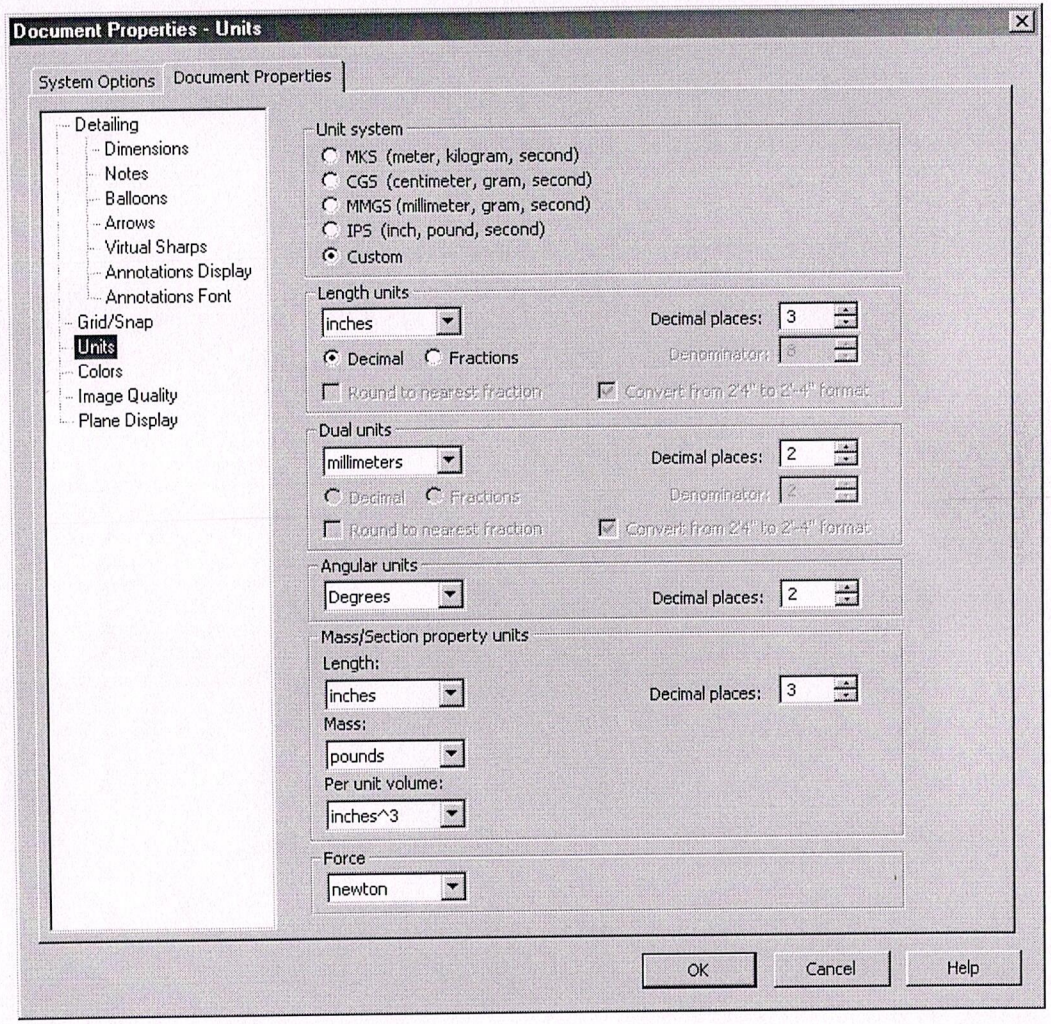


Document Templates

- Document Templates control the units, grid, text, and other settings for the model.
- The Tutorial document templates are required to complete the exercises in the *Online Tutorials*.
- The templates are located in the Tutorial tab on the New SolidWorks Document dialog box.
- Document properties are saved in templates.

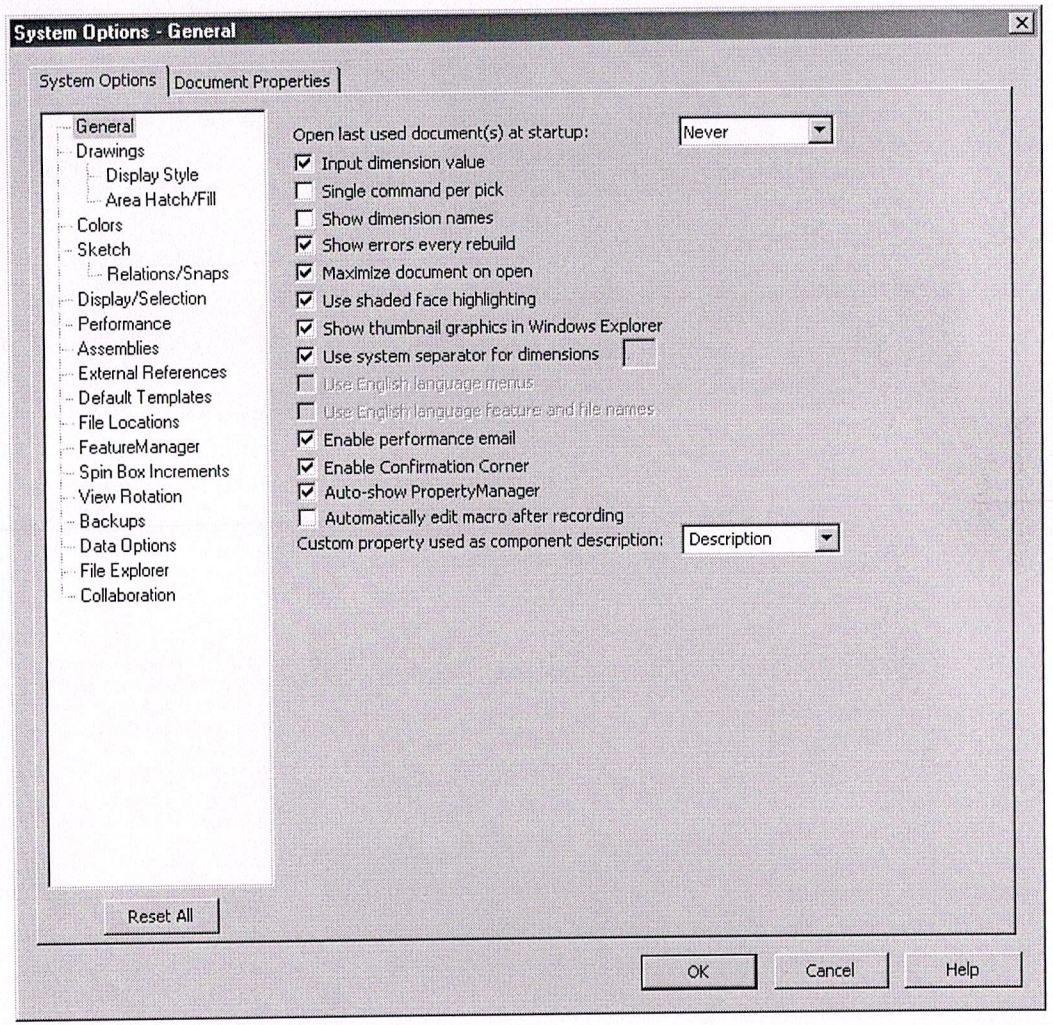
Document Properties

- **Accessed through the Tools, Options menu.**
- **Control settings like:**
  - **Units: English (inches) or Metric (millimeters)**
  - **Grid/Snap Settings**
  - **Colors, Material Properties and Image Quality**



System Options

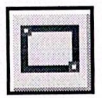
- Accessed through the Tools, Options menu.
- Allow you to customize your work environment.
- System options control:
  - File locations
  - Performance
  - Spin box increments



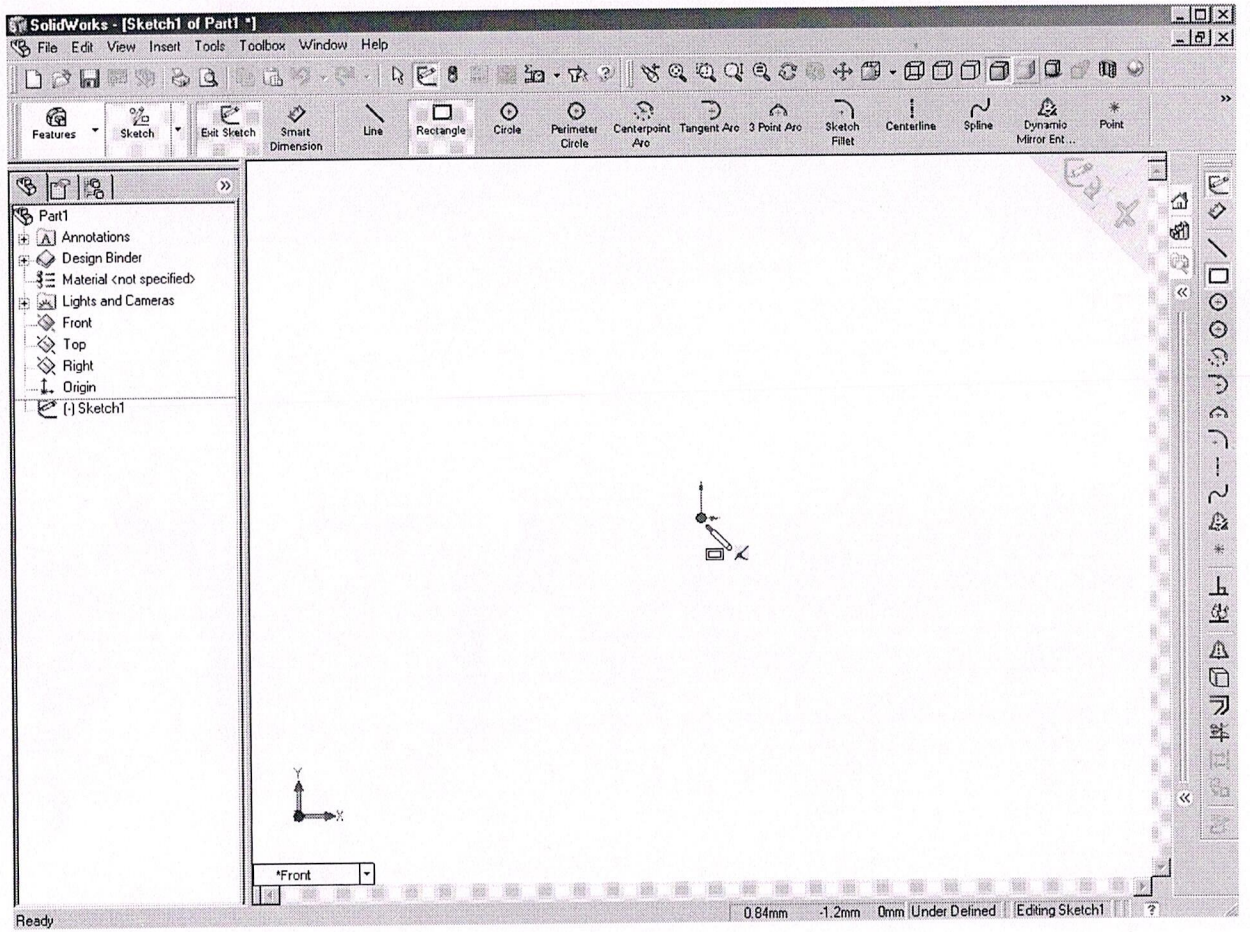
Creating a 2D Sketch

1. Click Sketch  the Sketch toolbar.

2. Select the Front plane as a sketch plane.

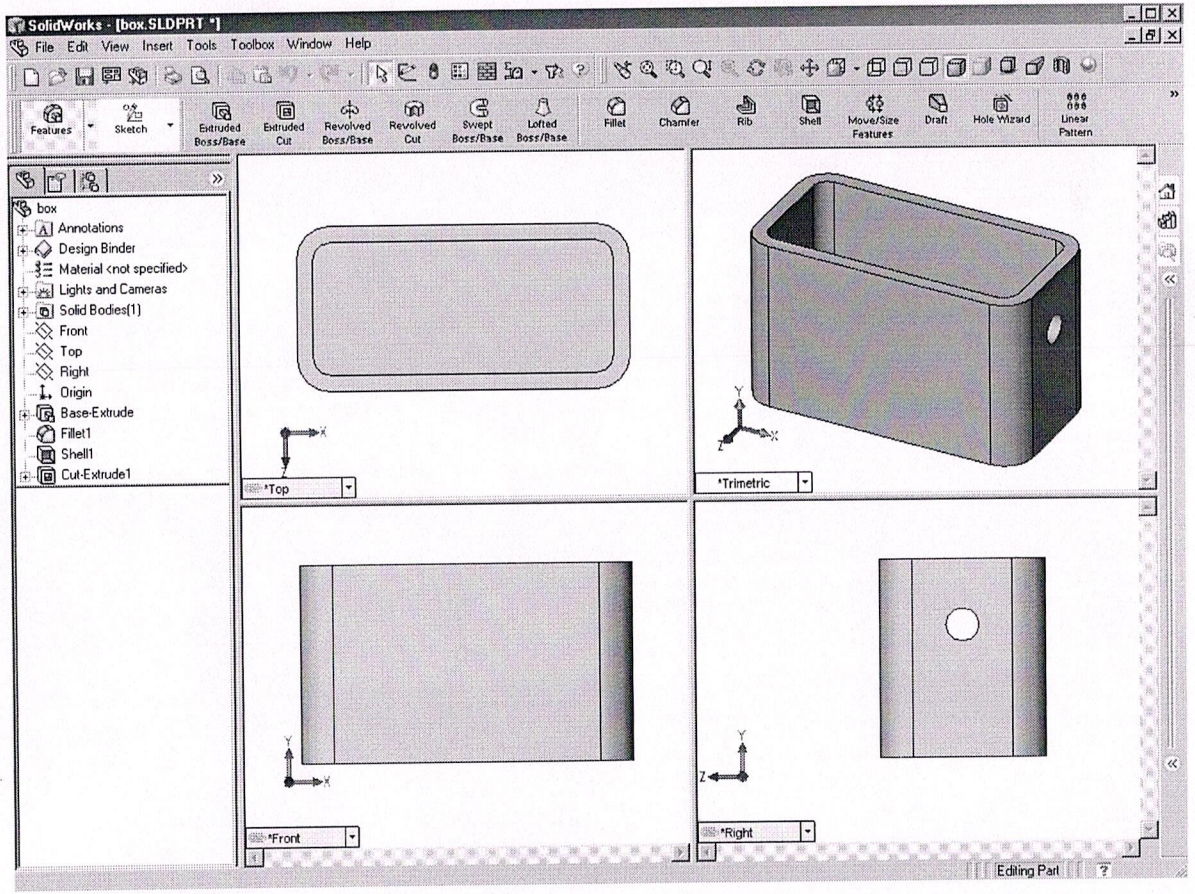
3. Click Rectangle  on the Sketch Tools toolbar.

4. Move the pointer to the Sketch Origin.



Multiple Views of a Document

- **Click the view pop-up menu.**
- **Select an icon. The viewport icons include:**
  - Single View
  - Two View (horizontal and vertical)
  - Four View

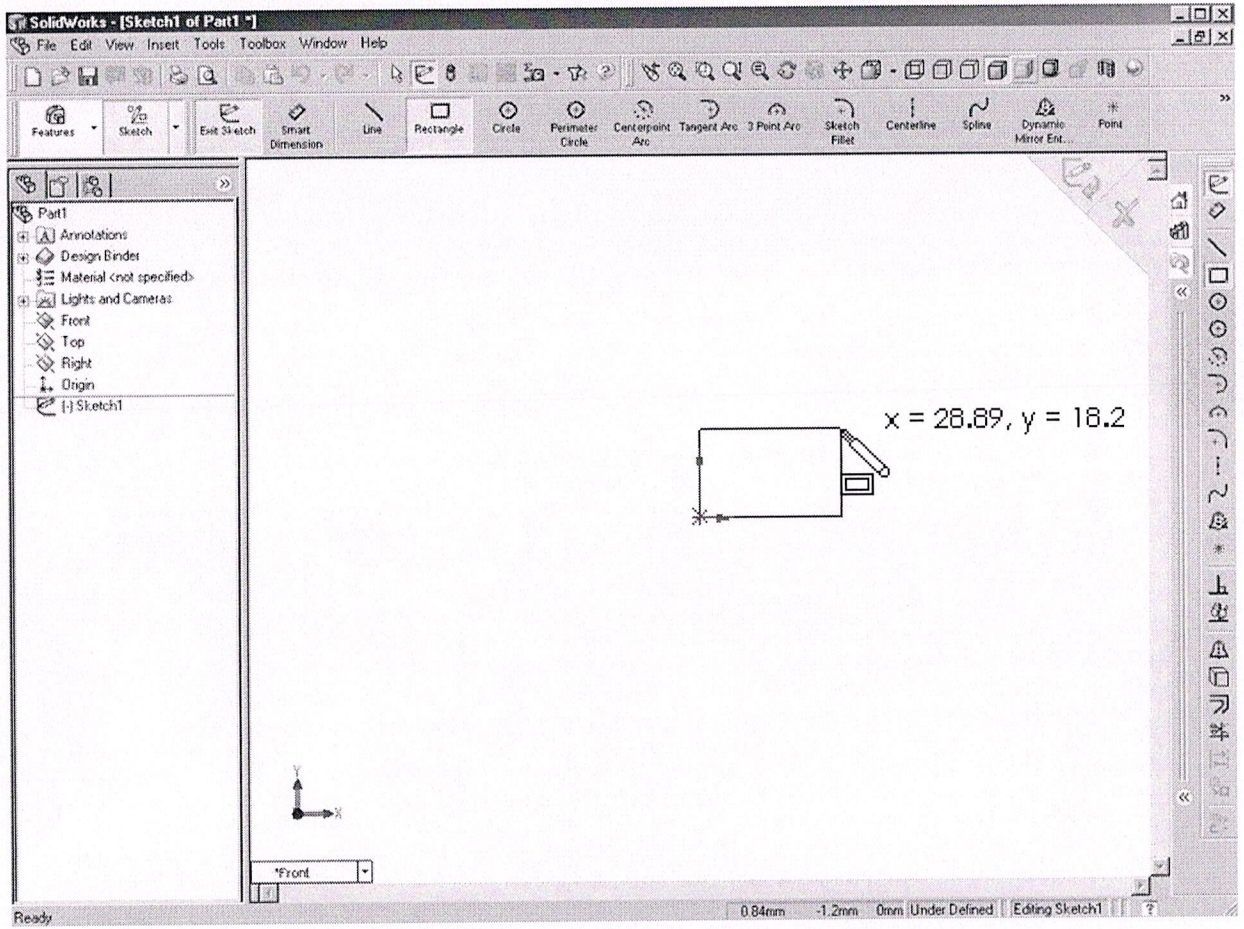


Creating a 2D Sketch

5. **Click the left mouse button.**

6. **Drag the pointer up and to the right.**

7. **Click the left mouse button again.**



Adding Dimensions

- **Dimensions specify the size of the model.**

To create a dimension:

1. **Click Dimension on the Sketch Relations toolbar.**
2. **Click the 2D geometry.**
3. **Click the text location.**
4. **Enter the dimension value.**