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

"CATIA V5 R21"

Resource Persons : 1. Sri B.SRIHARI Asst. Professor, Dept. of ME, KSRMCE

Course Coordinator : 1.Sri C. Naga Raja, Asst. Professor, Dept. of ME, KSRMCE

Date: 30/05/22 to 16/06/22

Lr./KSRMCE/ME/2021-22/ Date: 28-05-2022

To
The Principal,
KSRMCE,
Kadapa.

Sub: Permission to Conduct Certificate Course on "CATIA V5 R21" from 30-05-2022 to 16-06-2022 – Reg.

Respected Sir,

The Department of Mechanical Engineering is planning to offer a certification course on "CATIA V5 R21" to B. Tech. VI Semester students. The course will be conducted from 30-05-2022 to 16-06-2022. In this regard, we are requesting you to grant permission to conduct certificate course.

Thanking you

Yours faithfully
Sri C.NagaRaja, Asst. Professor

Forwarded to Principal S.V.

Permitted
V.S.S. Murthy

Cr./KSRMCE/ME/2021-22/ Date: 28/05/2022

Circular

The Department of Mechanical Engineering is offering a certification course on "**CATIA V5 R21**" From 30-05-2022 to 16-06-2022 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

Sri C.NagaRaja, Asst. Professor

Sri B.SRIHARI Asst. Professor

Dept. of Mechanical Engg.-K.S.R.M.CE.

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

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Department of Mechanical Engineering
Certification Course on CATIA V5 R21

List of Participants

S.no	Roll No	Name of the Student	Email Id's
1	199Y1A0302	B SRINIDHI SAI	199Y1A0302@ksrmce.ac.in
2	199Y1A0303	BANDI SHIVA REDDY	199Y1A0303@ksrmce.ac.in
3	199Y1A0304	CHAGANTI SUNIL KUMAR REDDY	199Y1A0304@ksrmce.ac.in
4	199Y1A0305	CHEPPALI AMATHYA	199Y1A0305@ksrmce.ac.in
5	199Y1A0306	CHIRUCHAPALA ABDUL SUBAHAN	199Y1A0306@ksrmce.ac.in
6	199Y1A0307	DEVAPATLA BHARATH SIMHA REDDY	199Y1A0307@ksrmce.ac.in
7	199Y1A0308	DUDIMANI SAI SRUJAN KUMAR	199Y1A0308@ksrmce.ac.in
8	199Y1A0309	GADIKOTA MURALIDHAR REDDY	199Y1A0309@ksrmce.ac.in
9	199Y1A0310	GANGALA VENKATA PRATHAP	199Y1A0310@ksrmce.ac.in
10	199Y1A0311	GANUGAPENTA BHARATH	199Y1A0311@ksrmce.ac.in
11	199Y1A0312	GODDENDLA ASHOK KUMAR	199Y1A0312@ksrmce.ac.in
12	199Y1A0313	GUDURU SUBHAN	199Y1A0313@ksrmce.ac.in
13	199Y1A0315	KAPURAM VAMSINATH REDDY	199Y1A0315@ksrmce.ac.in
14	199Y1A0316	KETHIREDDY NAVEEN KUMAR REDDY	199Y1A0316@ksrmce.ac.in
15	199Y1A0317	KONANGI SUBBANNA	199Y1A0317@ksrmce.ac.in
16	199Y1A0318	KONERU SAI KARTHIK	199Y1A0318@ksrmce.ac.in
17	199Y1A0319	KOTHAPALLE VAMSIDHAR REDDY	199Y1A0319@ksrmce.ac.in
18	199Y1A0320	KUMMARI MANJUNATH	199Y1A0320@ksrmce.ac.in

19	199Y1A0321	KUMMETHA SAI KUMAR REDDY	199Y1A0321@ksrmce.ac.in
20	199Y1A0322	L M VINAY KUMAR	199Y1A0322@ksrmce.ac.in
21	199Y1A0324	MALEPATI SIVA SAI REDDY	199Y1A0324@ksrmce.ac.in
22	199Y1A0325	MANJUNATHA DINESH KUMAR	199Y1A0325@ksrmce.ac.in
23	199Y1A0326	MARKAPURAM MYSORA REDDY	199Y1A0326@ksrmce.ac.in
24	199Y1A0327	MEDIMALA KIRAN KUMAR	199Y1A0327@ksrmce.ac.in
25	199Y1A0328	MOGHAL JUNAID BAIG	199Y1A0328@ksrmce.ac.in
26	199Y1A0329	MOLAKALA SREEKANTH REDDY	199Y1A0329@ksrmce.ac.in
27	199Y1A0330	MOYILLA CHARAN REDDY	199Y1A0330@ksrmce.ac.in
28	199Y1A0331	NADIMINTI NAVANEETH KUMAR	199Y1A0331@ksrmce.ac.in
29	199Y1A0332	NAGULAGARI VENKATA SANDEEP KUMAR REDDY	199Y1A0332@ksrmce.ac.in
30	199Y1A0333	PAGIDIPALEM VAMSI ACHARI	199Y1A0333@ksrmce.ac.in
31	199Y1A0334	PALLETI VAMSIDHAR REDDY	199Y1A0334@ksrmce.ac.in
32	199Y1A0335	PASUPALA RAVI KUMAR	199Y1A0335@ksrmce.ac.in
33	199Y1A0336	PATHAN KHALEELULLA KHAN	199Y1A0336@ksrmce.ac.in
34	199Y1A0337	PULAKONDAM BHEEMAIAH	199Y1A0337@ksrmce.ac.in
35	199Y1A0338	REDDAM VEERA TEJASWAR REDDY	199Y1A0338@ksrmce.ac.in
36	199Y1A0339	S K RAJESH	199Y1A0339@ksrmce.ac.in
37	199Y1A0340	SAGIRAJU DILLI VARMA	199Y1A0340@ksrmce.ac.in
38	199Y1A0341	SHAIK ABDUL RASHEED	199Y1A0341@ksrmce.ac.in
39	199Y1A0343	SHAIK GHOUSE BASHA	199Y1A0343@ksrmce.ac.in
40	199Y1A0344	SHAIK KURNOOL DADA KHALANDAR	199Y1A0344@ksrmce.ac.in
41	199Y1A0345	SHAIK MAHAMMED MANSOOR	199Y1A0345@ksrmce.ac.in
42	199Y1A0346	SHAIK MOHAMMED GHOUSE	199Y1A0346@ksrmce.ac.in
43	199Y1A0347	SHAIK MOHAMMED SAJID	199Y1A0347@ksrmce.ac.in
44	199Y1A0348	SHAIK MOHAMMED SHOAIB AKTHAR	199Y1A0348@ksrmce.ac.in
45	199Y1A0349	SHAIK NAYEEMUR RAHMAN	199Y1A0349@ksrmce.ac.in
46	199Y1A0350	SHAIK ZABEEULLA	199Y1A0350@ksrmce.ac.in
47	199Y1A0352	SUDA ABHILASH KUMAR REDDY	199Y1A0352@ksrmce.ac.in

48	199Y1A0354	SYED ASLAM	199Y1A0354@ksrmce.ac.in

Coordinator

HOD

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

Syllabus of Certification Course

Course Name: CATIA V5 R21

Duration: 30 Hours

Module-1 10 Hrs

Sketching: Outline of course, Overview of the CATIA V5 User Interface and suggested settings. Drawing Templates and Sheet Formats : Default Drawing Templates, Sheet Format and Sheet Size, New catia 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 Modeling

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

Text Books:

1. K.L.Narayana, P.Kannaiah ,Machine Drawing ,New age international Publications, sixthEdition 2019
2. Dr.R.K Dahwan ,A Text Book of Machine Drawing ,s.chand Publications,2018

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

SCHEDULE

Department of Mechanical Engineering

Certification course
on
"CATIA V5 R21"

Date	Timing	Course Instructor	Topic to be covered
30/05/2022	4 PM to 6 PM	B.SRIHARI	INTRODUCTION TO CATIA V5 R21 WORKS SKETCHING: Outline of course, Overview of the CATIA V5 R21 Works User Interface and suggested settings.
31/05/2022	4 PM to 6 PM	B.SRIHARI	Drawing Templates and Sheet Formats : Default Drawing Templates, Sheet Format and Sheet Size, New CATIA V5 R21Works Document, Sheet Format/Size .Drawing Sheet Size and Format. Draw tools, Editing Tool
01/06/2022	4 PM to 6 PM	B.SRIHARI	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,
02/06/2022	4 PM to 6 PM	B.SRIHARI	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
03/06/2022	4 PM to 6 PM	B.SRIHARI	Dimensioning - Smart, Horizontal, Vertical, Ordinate, Horizontal ordinate, Vertical ordinate, Align ordinate, Fully define sketch. Sketch Diagnosis, Sketch expert, 3D Sketching, Rapid Sketch and EXERCISES
04/06/2022	4 PM to 6 PM	B.SRIHARI	Part Modeling: Part Modeling Tools Creating reference planes Creating Extrude features –

			Direction1, Direction2, From option, Thin feature, Applying draft, Selecting contours
06/06/2022	4 PM to 6 PM	B.SRIHARI	Creating Revolve features – Selecting Axis, Thin features, Selecting contours Creating Swept features-Selecting, Profile and Path,
07/06/2022	4 PM to 6 PM	B.SRIHARI	Orientation/twist type, Path Alignment, Guide Curves, Start/End tangency, Thin feature
08/06/2022	4 PM to 6 PM	B.SRIHARI	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
09/06/2022	4 PM to 6 PM	B.SRIHARI	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
10/06/2022	4 PM to 6 PM	B.SRIHARI	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
13/06/2022	4 PM to 6 PM	B.SRIHARI	Applying Advanced Mates – Symmetric, Width, Path Mate, Linear/Linear Coupler, Limit Mate. Applying Smart mates Applying Mate reference
14/06/2022	4 PM to 6 PM	B.SRIHARI	Manipulating Components - Replacing Components, Rotating Components, Move Components, Collision Detection, Physical Dynamics, Dynamic Clearance, Detecting Interference Creating Pattern - Assembly Pattern,
15/06/2022	4 PM to 6 PM	B.SRIHARI	Mirror Creating Explode Views Top Down Design – Layout Sketch, Work Part In the Context of an assembly. Smart Components, Smart Fasteners, Physical Simulation
16/06/2022	4 PM to 6 PM	B.SRIHARI	Exercises and Conclusion

Resource persons:

Coordinators:

HOD

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

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Kadapa, Andhra Pradesh, India–516 005

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Department of Mechanical Engineering

Certification Course on CATIA V5 R21

Attendance Sheet

S. No	Name of the Student	Roll List	30/05	31/05	01/06	02/06	03/06	04/06	6/6	07/06	08/06	09/06	10/06	13/06	14/0	15/06	16/06
1	B SRINIDHI SAI	199Y1A0302	P	P	A	P	P	P	P	P	P	P	P	A	P	P	P
2	BANDI SHIVA REDDY	199Y1A0303	P	P	P	P	P	P	A	P	P	A	P	P	A	P	P
3	CHAGANTI SUNIL KUMAR REDDY	199Y1A0304	P	P	P	P	P	A	P	A	P	P	P	P	P	A	P
4	CHEPPALI AMATHYA	199Y1A0305	P	A	P	P	P	P	P	P	P	P	A	P	P	P	P
5	CHIRUCHAPALA ABDUL SUBAHAN	199Y1A0306	P	P	P	P	P	P	P	A	A	P	P	A	P	P	P
6	DEVAPATLA BHARATH SIMHA REDDY	199Y1A0307	P	P	P	P	A	P	P	A	P	P	P	P	A	P	P
7	DUDIMANI SAI SRUJAN KUMAR	199Y1A0308	P	P	P	P	P	P	A	P	P	A	P	P	P	P	P
8	GADIKOTA MURALIDHAR REDDY	199Y1A0309	P	P	P	A	P	P	P	P	A	P	P	P	P	A	P
9	GANGALA VENKATA PRATHAP	199Y1A0310	P	P	A	P	P	P	P	P	P	P	P	A	P	P	A

10	GANUGAPENTA BHARATH	199Y1A0311	P	P	P	P	A	P	A	P	P	P	P	A	P	A	P
11	GODDENDLA ASHOK KUMAR	199Y1A0312	P	P	P	P	P	P	P	P	A	A	P	P	P	P	A
12	GUDURU SUBHAN	199Y1A0313	A	P	P	P	A	P	P	P	P	A	P	P	P	P	P
13	KAPURAM VAMSINATH REDDY	199Y1A0315	P	P	P	P	P	P	A	P	P	P	A	P	P	P	P
14	KETHIREDDY NAVEEN KUMAR REDDY	199Y1A0316	P	P	P	P	P	A	P	P	P	P	P	P	A	P	P
15	KONANGI SUBBANNA	199Y1A0317	P	P	P	P	P	P	P	P	P	P	A	P	P	P	P
16	KONERU SAI KARTHIK	199Y1A0318	P	A	P	P	A	P	P	P	P	P	P	P	P	P	P
17	KOTHAPALLE VAMSIDHAR REDDY	199Y1A0319	P	P	P	A	P	P	P	A	P	P	P	A	P	P	P
18	KUMMARI MANJUNATH	199Y1A0320	P	P	P	P	P	P	P	P	A	P	P	P	P	A	P
19	KUMMETHA SAI KUMAR REDDY	199Y1A0321	P	P	P	A	P	P	A	P	P	A	P	P	P	P	P
20	L M VINAY KUMAR	199Y1A0322	P	P	A	P	P	P	P	P	P	P	P	P	P	P	P
21	MALEPATI SIVA SAI REDDY	199Y1A0324	P	P	P	P	P	A	P	P	A	P	A	P	P	P	P
22	MANJUNATHA DINESH KUMAR	199Y1A0325	P	P	P	P	P	P	P	A	P	P	P	P	P	P	A
23	MARKAPURAM MYSORA REDDY	199Y1A0326	P	P	P	P	A	P	P	P	P	P	P	A	P	P	P
24	MEDIMALA KIRAN KUMAR	199Y1A0327	P	P	P	P	P	P	P	P	P	P	A	P	P	P	P
25	MOGHAL JUNAID BAIG	199Y1A0328	A	P	P	P	P	P	P	P	P	P	P	A	P	P	P
26	MOLAKALA SREEKANTH REDDY	199Y1A0329	P	P	P	P	P	P	P	A	P	A	P	P	A	P	P
27	MOYILLA CHARAN REDDY	199Y1A0330	P	P	P	A	P	P	P	P	P	P	P	P	P	P	P
28	NADIMINTI NAVANEETH	199Y1A0331	P	A	P	P	P	P	P	P	P	P	P	A	P	P	P

	KUMAR															A	
29	NAGULAGARI VENKATA SANDEEP KUMAR REDDY	199Y1A0332	P	A	P	A	P	P	P	P	P	P	P	A	P	P	P
30	PAGIDIPALEM VAMSI ACHARI	199Y1A0333	P	P	A	P	P	P	P	P	P	P	A	P	P	P	A
31	PALLETI VAMSIDHAR REDDY	199Y1A0334	P	P	P	P	P	A	P	A	P	P	P	P	P	P	P
32	PASUPALA RAVI KUMAR	199Y1A0335	P	P	P	P	P	P	A	P	P	P	P	P	P	P	
33	PATHAN KHALEELULLA KHAN	199Y1A0336	A	P	P	P	P	P	P	P	P	P	A	P	A	P	A
34	PULAKONDAM BHEEMAIAH	199Y1A0337	P	P	P	P	A	P	P	P	P	P	P	P	P	P	A
35	REDDAM VEERA TEJASWAR REDDY	199Y1A0338	P	P	P	P	P	P	A	P	P	P	P	P	A	P	P
36	S K RAJESH	199Y1A0339	P	A	P	P	P	A	P	P	P	A	P	P	P	P	P
37	SAGIRAJU DILLI VARMA	199Y1A0340	P	P	P	A	P		P	A		P	P	P	P	P	P
38	SHAIK ABDUL RASHEED	199Y1A0341	P	P	P	P	P	A	P	P	A	P	P	P	P	P	P
39	SHAIK GHOUSE BASHA	199Y1A0343	P	P	A	P	P	P	P	P	P	P	A	P	P	A	P
40	SHAIK KURNOOL DADA KHALANDAR	199Y1A0344	A	P	P	A	P	P	P	P	P	P	P	P	P	P	P
41	SHAIK MAHAMMED MANSOOR	199Y1A0345	P	P	P	P	P	P	P	A	P	A	P	P	A	P	P
42	SHAIK MOHAMMED GHOUSE	199Y1A0346	P	P	P	P	P	P	A	P	P	P	A	P	P	P	A
43	SHAIK MOHAMMED SAJID	199Y1A0347	A	P	P	P	A	P	P	P	P	P	P	A	P	P	P
44	SHAIK MOHAMMED SHOAIB AKTHAR	199Y1A0348	P	P	P	P	P	P	P	P	A	P	P	P	A	P	A
45	SHAIK NAYEEMUR RAHMAN	199Y1A0349	P	P	P	P	P	A	A	P	P	P	A	P	P	P	P
46	SHAIK ZABEEULLA	199Y1A0350	P	P	A	P	P	P	P	P	P	P	P	P	P	P	P

47	SUDA ABHILASH KUMAR REDDY	199Y1A0352	P	P	P	P	A	P	P	A	P	P	P	P	P	A	P
48	SYED ASLAM	199Y1A0354	P	P	P	P	P	P	P	P	A	P	A	P	P	P	P

Coordinator

HOD

Professor & Head
Department of Mechnical Engineering
K.S.R.M. College of Engineering
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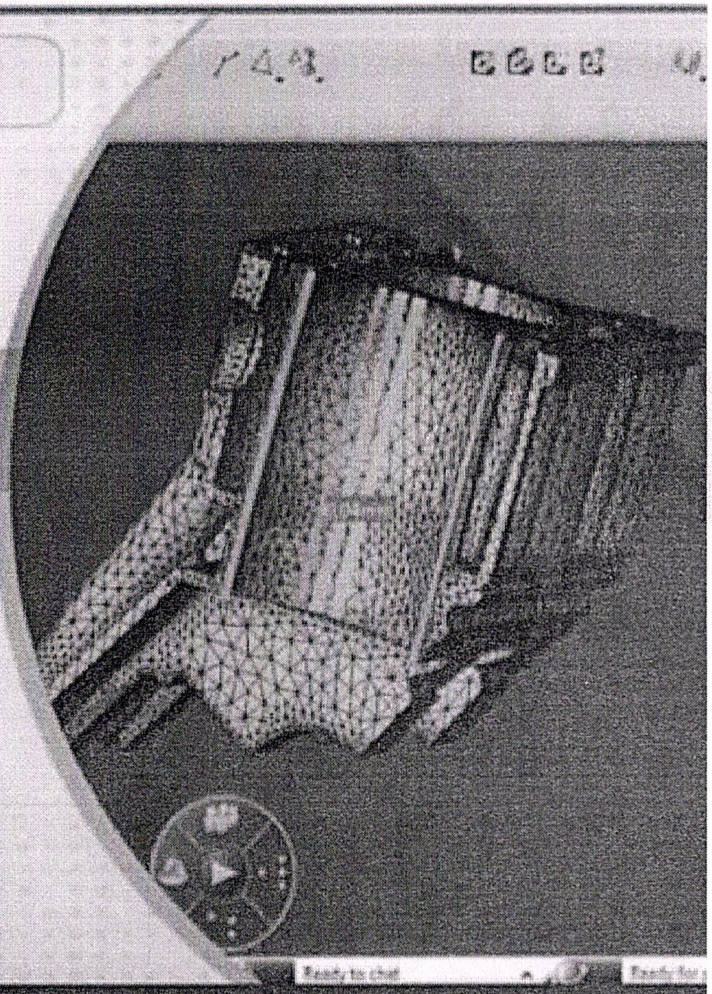
DEPARTMENT OF MECHANICAL ENGINEERING



CERTIFICATION COURSE ON

" CATIA V5R21 "

30 - 05 - 2022 TO 16 - 06 - 2022

ME 206

Coordinator

C.NAGARAJA

Assistant Professor,

Ksrmce.ME dept.

Resource person

B.SRIHARI

Assistant Professor,

Ksrmce.ME dept.

Report of
Certification Course on "CATIA V5 R21"
From 30/05/2022 to 16/06/2022

Target Group	:	**VI Semester Students**
Details of Participants	:	**48 Students**
Co-coordinator(s)	:	**C. Nagaraja**
Resource Persons	:	**B. Srihari**
Organizing Department	:	**Mechanical Engineering**
Venue	:	**Engineering Graphics Lab, Mechanical Department**

Description:

The Department of Mechanical Engineering conducted a certification course on "CATIA V5 R21" from 30th May 2022 to 16th June 2022. The course duration was 30 hours .The course Resource Persons are Sri B. Srihari, Assistant Professor Department Mechanical Engineering, KSRMCE.

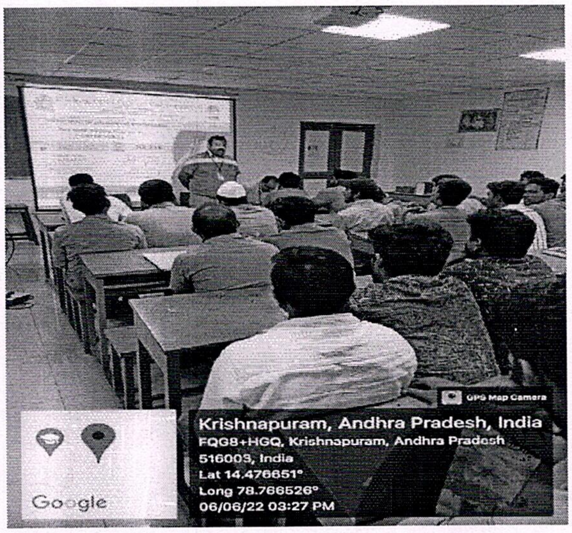
CATIAV5 R21 is an easy to use parametric design modular, meaning you can easily edit the design at any stage in the design process. Real View graphics allow you to visualize your design in real time whilst Photo View 360 can create sophisticated photo realistic renderings and animations. Both tools will 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.

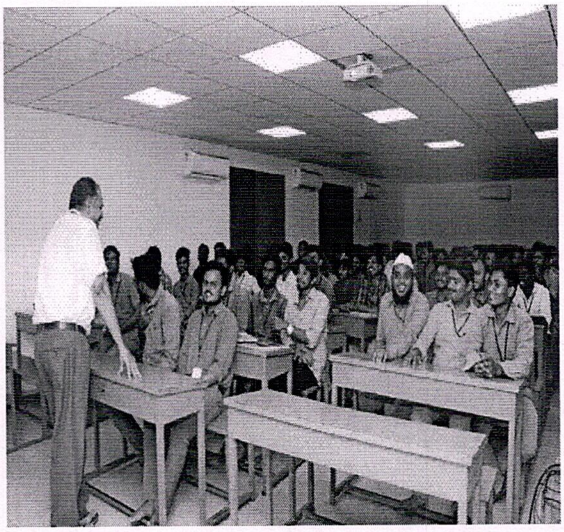
" CATIA V5 R21 is the preeminent software for computer-aided design and computer-aided engineering. Multiple parts can be combined together into assemblies by assigning relationships (called "mates") among various features. Both assemblies and individual parts can be turned into engineering drawings 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. "

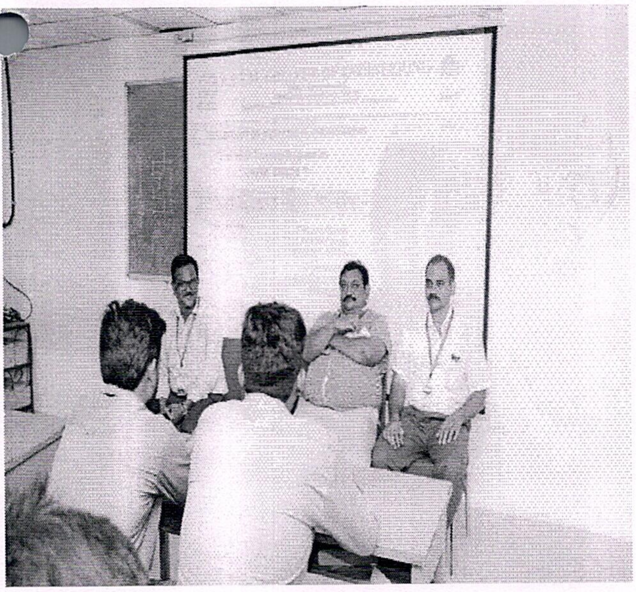
At the End of the course Students will able to do 2D modeling, Part Modeling and Assembly of components and able to Model Industry standard Mechanical Drawings.

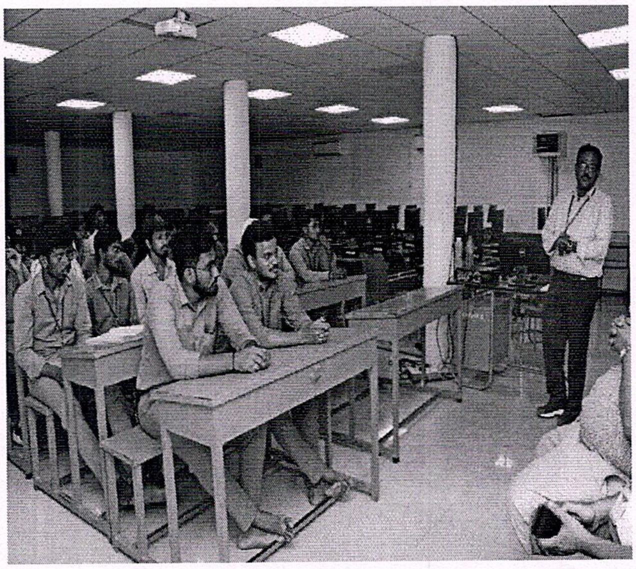
Photos

The pictures taken during the course are given below:









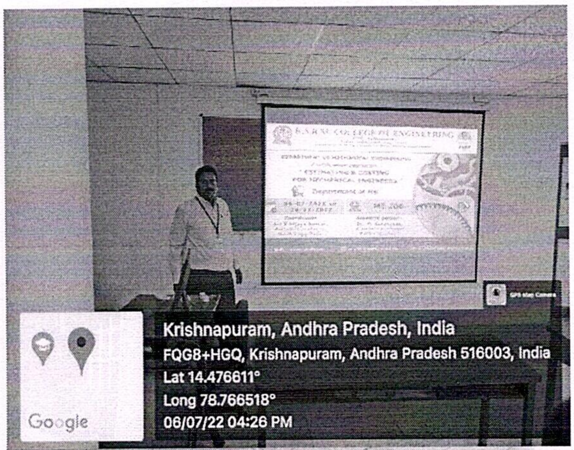
Coordinators

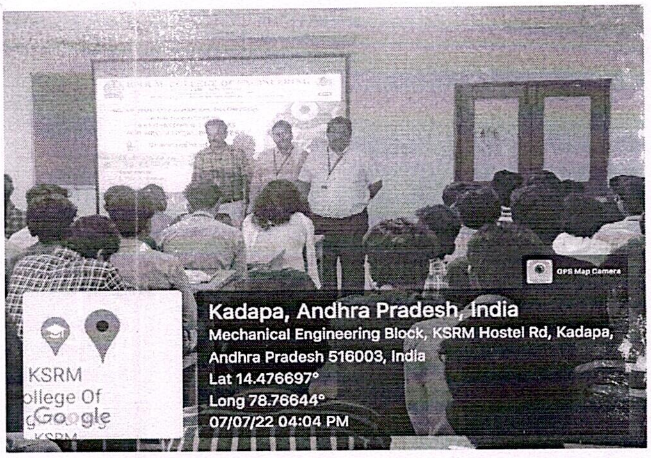
HOD

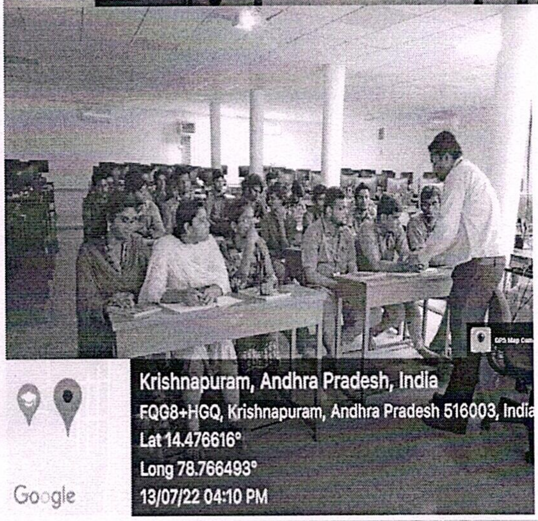
Professor & Head
Department of Mechnical Engineering
K.S.R.M. College of Engineering
KADAPA - 516 003.

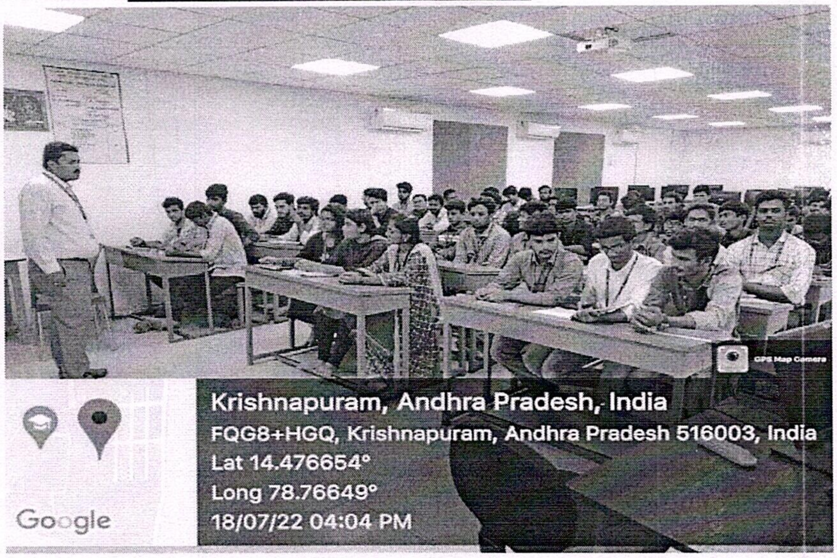
Photos

The pictures taken during the course are given below:









S. Vijay Kumar

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.

LIGHTED TO LIGHTEN

KSNR
lives on..

Certificate of Completion

This to certify that Mr/Mrs. SK RAJESH Bearing the Roll Number 1994IA0339 has Successfully Completed Certification Course on " CATIA V5 R21 " from 30-05-2022 to 16-06-2022, Organized by Department of Mechanical Engineering, KSRMCE, Kadapa.

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.

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Certificate of Completion

This to certify that Mr/Mrs. BANDI SHIVA REDDY Bearing the Roll Number 199Y1A0303 has Successfully Completed Certification Course on " CATIA V5 R21 " from 30-05-2022 to 16-06-2022 ,Organized by Department of Mechanical Engineering, KSRMCE, Kadapa.

Coordinator

HOD ME

V. S. S. Murthy
Principal

PRINCIPAL
K.S.R.M. COLLEGE OF ENGINEERING
KADAPA - 516 003.

K.S.R.M. COLLEGE OF ENGINEERING

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Kadapa, Andhra Pradesh, India– 516 003
Approved by AICTE, New Delhi & Affiliated to JNTUA, Ananthapuramu.

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Certificate of Completion

This to certify that Mr/Mrs. S. MOHAMMED GHOUSE Bearing the Roll Number 19941A0346 has Successfully Completed Certification Course on " CATIA V5 R21 " from 30-05-2022 to 16-06-2022 ,Organized by Department of Mechanical Engineering, KSRMCE, Kadapa.

Coordinator

HOD ME

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

V. S. S. Murthy
Principal

PRINCIPAL
K.S.R.M. COLLEGE OF ENGINEERING

Feedback on Certificate Course on "CATIA V5 R21" From 30/05/2022 to 16/06/2022

*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

This content is neither created nor endorsed by Google.

Google Forms

Feedback on Certificate Course on" CATIA V5 R21 " from 30/05/22 to 16/06/22

S.No	Timestamp	The object	The pa	The content	The Resource person	The exercisces	The venu	The Course	Student Nam	Roll Number(C	Any Other comments
1	16-06-2022 17:10:20	Excellent	Excellent	Excellent	Excellent	Excellent	Excellent	Excellent			
2	16-06-2022 17:10:25	Excellent	Good	Excellent	Good	Good	Excellent	Excellent	Mysore Reddy	199Y1A0326	Good Session
3	16-06-2022 17:10:32	Excellent	Excellent	Excellent	Excellent	Excellent	Excellent	Excellent			
4	16-06-2022 17:10:33	Good	Excellent	Good	Good	Good	Good	Good			--
5	16-06-2022 17:10:35	Excellent	Excellent	Excellent	Good	Excellent	Excellent	Excellent			
6	16-06-2022 17:10:42	Excellent	Excellent	Excellent	Excellent	Excellent	Excellent	Excellent	Siva Reddy	199Y1A0303	
7	16-06-2022 17:10:44	Excellent	Excellent	Excellent	Excellent	Excellent	Good	Good			We need some more EXAMPLES
8	16-06-2022 17:10:47	Excellent	Excellent	Excellent	Good	Excellent	Good	Good			
9	16-06-2022 17:10:51	Satisfactory	Satisfactory	Satisfactory	Satisfactory	Satisfactory	Satisfactory	Satisfactory			No
10	16-06-2022 17:10:52	Excellent	Good	Excellent	Good	Good	Excellent	Excellent	Sreekanth	199Y1A0329	
11	16-06-2022 17:10:55	Excellent	Excellent	Excellent	Excellent	Excellent	Excellent	Excellent			
12	16-06-2022 17:10:56	Good	Excellent	Good	Good	Good	Good	Good			
13	16-06-2022 17:10:58	Excellent	Excellent	Excellent	Excellent	Excellent	Excellent	Excellent			
14	16-06-2022 17:11:01	Excellent	Excellent	Good	Good	Good	Good	Good			It is useful for us
15	16-06-2022 17:11:10	Excellent	Good	Excellent	Good	Good	Excellent	Excellent			--
16	16-06-2022 17:11:15	Excellent	Excellent	Excellent	Excellent	Excellent	Excellent	Excellent			
17	16-06-2022 17:11:17	Excellent	Excellent	Excellent	Excellent	Excellent	Excellent	Excellent			
18	16-06-2022 17:11:20	Excellent	Good	Excellent	Good	Excellent	Excellent	Good	Vamsi	199Y1A0319	i Learned something new
19	16-06-2022 17:11:22	Excellent	Excellent	Excellent	Excellent	Good	Good	Good			
20	16-06-2022 17:11:25	Good	Good	Good	Excellent	Good	Good	Excellent			
21	16-06-2022 17:11:27	Excellent	Excellent	Good	Excellent	Good	Excellent	Good	Naveen	199Y1A0317	
22	16-06-2022 17:11:30	Good	Good	Good	Good	Good	Good	Good			--
23	16-06-2022 17:11:32	Excellent	Good	Good	Satisfactory	Excellent	Excellent	Excellent			
24	16-06-2022 17:11:35	Excellen	Excellent	Excellen	Excellen	Excellent	Excellent	Excellent			-

		t		t	t						
25	16-06-2022 17:11:46	Excellent	Good	Excellent	Good	Good	Excellent	Excellent			Nothing to say
26	16-06-2022 17:11:54	Excellent	Excellent	Excellent	Excellent	Excellent	Excellent	Excellent			
27	16-06-2022 17:12:02	Good	Excellent	Good	Good	Good	Good	Good			
28	16-06-2022 17:12:05	Excellent	Excellent	Excellent	Excellent	Excellent	Excellent	Excellent			--
29	16-06-2022 17:12:07	Excellent	Excellent	Good	Good	Good	Good	Good			
30	16-06-2022 17:12:19	Excellent	Good	Excellent	Good	Good	Excellent	Excellent	vamshi	199Y1A0315	
31	16-06-2022 17:12:20	Excellent	Excellent	Excellent	Excellent	Excellent	Excellent	Excellent			
32	16-06-2022 17:12:24	Good	Good	Excellent	Good	Good	Good	Good			
33	16-06-2022 17:13:20	Excellent	Excellent	Good	Good	Good	Good	Good			
34	16-06-2022 17:13:34	Excellent	Excellent	Excellent	Excellent	Excellent	Excellent	Excellent			-
35	16-06-2022 17:13:42	Good	Good	Good	Good	Good	Good	Good			No
36	16-06-2022 17:13:48	Excellent	Good	Good	Good	Satisfactory	Excellent	Excellent			
37	16-06-2022 17:13:59	Good	Good	Good	Good	Good	Good	Good			No
38	16-06-2022 17:14:20	Excellent	Excellent	Good	Good	Good	Good	Good			
39	16-06-2022 17:14:02	Excellent	Excellent	Excellent	Excellent	Excellent	Excellent	Excellent			
40	16-06-2022 17:14:10	Good	Good	Good	Good	Good	Good	Good			
41	16-06-2022 17:14:20	Good	Good	Good	Good	Good	Good	Good			
42	16-06-2022 17:14:28	Excellent	Excellent	Excellent	Excellent	Excellent	Excellent	Excellent			
43	16-06-2022 17:15:42	Good	Good	Good	Good	Good	Good	Good			
44	16-06-2022 17:15:45	Good	Good	Good	Good	Good	Good	Good			--
45	16-06-2022 17:15:51	Excellent	Excellent	Excellent	Excellent	Excellent	Excellent	Excellent		199Y1A0335	
46	16-06-2022 17:16:53	Good	Good	Good	Good	Good	Good	Good			
47	16-06-2022 17:16:55	Excellent	Excellent	Good	Good	Excellent	Excellent	Good			Nothing
48	16-06-2022 17:16:59	Good	Good	Good	Good	Good	Good	Good			

certification courses

K.S.R.M. COLLEGE OF ENGINEERING (AUTONOMOUS), KADAPA-516003
DEPARTMENT OF MECHANICAL ENGINEERING
VALUE ADDED COURSE ON
CATIA V5 R21
FROM 30/05/2022 TO 16/06/2022
AWARD LIST

S.No	Roll Number	Name of the Student	Marks Obtained
1	199Y1A0302	B Srinidhi Sai	14
2	199Y1A0303	Bandi Shiva Reddy	13
3	199Y1A0304	Chaganti Sunil Kumar Reddy	14
4	199Y1A0305	Cheppali Amathya	13
5	199Y1A0306	Chiruchapala Abdul Subahan	13
6	199Y1A0307	D. Bharath Simha Reddy	13
7	199Y1A0308	Dudimani Sai Srujn Kumar	14
8	199Y1A0309	Gadikota Muralidhar Reddy	13
9	199Y1A0310	Gangala Venkata Prathap	14
10	199Y1A0311	Ganugapenta Bharath	14
11	199Y1A0312	Goddendla Ashok Kumar	13
12	199Y1A0313	Guduru Subhan	14
13	199Y1A0315	Kapuram Vamsinath Reddy	12
14	199Y1A0316	K. Naveen Kumar Reddy	12
15	199Y1A0317	Konangi Subbanna	12
16	199Y1A0318	Koneru Sai Karthik	12
17	199Y1A0319	Kothapalle Vamsidhar Reddy	14
18	199Y1A0320	K. Manjunath	13
19	199Y1A0321	K Sai Kumar Reddy	14
20	199Y1A0322	L M Vinay Kumar	13
21	199Y1A0324	Malepati Siva Sai Reddy	13
22	199Y1A0325	M. Dinesh Kumar	13
23	199Y1A0326	M. Mysora Reddy	14
24	199Y1A0327	Medimala Kiran Kumar	13
25	199Y1A0328	M. Junaid Baig	14
26	199Y1A0329	M.Sreekanth Reddy	12
27	199Y1A0330	M. Charan Reddy	12
28	199Y1A0331	Nadiminti Navaneeth Kumar	14
29	199Y1A0332	Nagulagari Venkata Sandeep Kumar Reddy	13
30	199Y1A0333	Pagidipalem Vamsi Achari	14
31	199Y1A0334	P. Vamsidhar Reddy	12
32	199Y1A0335	P. Ravi Kumar	12
33	199Y1A0336	P. Khaleelulla Khan	14
34	199Y1A0337	P. Bheemaiah	12
35	199Y1A0338	R. Veera Tejaswar Reddy	12
36	199Y1A0339	S K Rajesh	13
37	199Y1A0340	Sagiraju Dilli Varma	14
38	199Y1A0341	Shaik Abdul Rasheed	12
39	199Y1A0343	Shaik Ghouse Basha	13
40	199Y1A0344	Shaik Kurnool Dada Khalandar	14

41	199Y1A0345	Shaik Mahammed Mansoor	12
42	199Y1A0346	Shaik Mohammed Ghouse	12
43	199Y1A0347	Shaik Mohammed Sajid	12
44	199Y1A0348	Shaikmohammed Shoaib Akthar	13
45	199Y1A0349	Shaik Nayeemur Rahman	14
46	199Y1A0350	Shaik Zabeeulla	13
47	199Y1A0352	Suda Abhilash Kumar Reddy	14
48	199Y1A0354	Syed Aslam	12

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
CATIA V5 R21 FROM 30/05/2022 TO 16/06/2022
ASSESSMENT TEST

13

Roll Number: 199Y1A0343 **Name of the Student:** S. Ghouse Basha

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

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

1. Basic Catia V5: A part in Catia V5 has which suffix? [A]
A. *.CATPart B. *.Model C. *.Prt D. *.Dwg

2. Basic Catia V5: Is "wrench-Bonn?et_string.CATPrpduct" an acceptable filename in Catia?
A. True B. False [B]

3. Basic Catia V5: If you are missing a toolbar, what do you do? [A]
A. Right-click on an icon and look for it there or check View-Toolbars
B. Start a different module of Catia and look for it there
C. Close and start Catia again
D. Close the open document and open it again

4. Basic Catia V5: An assembly in Catia V5 has which suffix? [D]
A. *.Model B. *.CATPart C. *.Prt D. *.CATProduct

5. Part Design: When a new plane is created, where, in the specification tree, will it appear? (It is NOT a hybrid model) [A]
A. In the Geometrical Set.1 body B. In the Part Body
C. Just after the zx plane

6. Part Design: What is a draft feature? [B]
A. The function is often used when the part is going to be cast/forged/formed somehow with a tool
B. It is a feature you can use for creating lines
C. The function is often used within styling
D. You can create a pad with fillets with this command

7. Part Design: The term "parametric" means that a solid geometry is controlled by driving dimensions.
A. True [B]
B. False

8. Part Design: When is it a good idea to use multi-body techniques? [A]
A. For instance when designing complex and/or molded parts
B. When assemble complex products
C. When you have many surfaces
D. When you need to create a drawing

9. Part Design: What is a skeleton? [A]
A. Help geometry that you can use as reference
B.A thin walled CATIA model
C. Skeleton is another name for a sketch
D. Skeleton is another name for a geometrical set.

10. Part Design: What is true about sketch-based features? [A]
A. You need at least one sketch to create them
B. They can only be created in the Part Body (the first body)
C. It is an isolated sketch
D. They need a surface as parent element

11. Sketcher: When work is finished, why is it a good idea to use Sketch analysis? [A]
A. To verify that geometry in sketch is correct
B. To measure geometry in sketch
C. To change geometry in sketch
D. To change color/thickness of geometry in sketch

12. Sketcher: The Sketcher workbench is a standalone workbench that allows you to save a *.CATSketch file [A]
A. True B. False

13. Drafting: How do you create your first view (Front view)? [B]
A. Pick front view icon and pick a plane in the 3D viewer, Part or Assembly
B. Copy the object from assembly or part and paste it on the drawing
C. Pick a plane on object and right click choose in menu Front view
D. Drag the part or assembly into the drawing sheet

14. Drafting: How do you see which drawing view is active? [C]
A. The active view has a red frame
B. All views are active!
C. Only the active view is visible

15. Drafting: When you create an exploded view, you usually use a tool called Enhanced Scene. Why?
A. You can create an exploded view which doesn't affect the "master" product [A]
B. You can create position balloons there
C. You can create a Bill of material list there
D. You can create an animated movie there

16. Drafting: In a detailed drawing, are the views dependant on the CATParts point of origin?
A. True
B. False [A]

17. Assembly Design: When analyzing a component, Catia says it has 6 "degrees of freedom". What is the meaning of that? [A
A. It means that the component is fully loose in space
B. It means that the component is fully fixed in space
C. It means that the component is not saved
D. It means that the component is drafted with an angle of 6-degrees

18. Assembly Design: What is the name of the command that makes it possible to create an exploded view? [A]
A. Enhanced Scene B. 3D split C. Move D. Assembly split

19. Surface Design: When shall you use the Join command? [B
A. When you want to join two or more surfaces or curves
B. When you want to join a solid body to a surface
C. When I need to connect a sketch to a surface
D. When I want to join a point to a curve

20. Surface Design: What is an isolated element? [D]
A. Element without a history, it has no relation to a parent element
B. Element with a fixed color
C. Element you only use in Part Design
D. Element with a specific history, you can only change the length parameter

K.S.R.M. COLLEGE OF ENGINEERING (AUTONOMOUS), KADAPA-516003
DEPARTMENT OF MECHANICAL ENGINEERING
VALUE ADDED /CERTIFICATE COURSE ON
CATIA V5 R21 FROM 30/05/2022 TO 16/06/2022
ASSESSMENT TEST

13

Roll Number: 199Y1A0305 Name of the Student: C. Amathya

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

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

1. Basic Catia V5: A part in Catia V5 has which suffix? [a]
A. *.CATPart B. *.Model C. *.Prt D. *.Dwg

2. Basic Catia V5: Is "wrench-Bonn?et_string.CATPrpduct" an acceptable filename in Catia?
A. True B. False [b]

3. Basic Catia V5: If you are missing a toolbar, what do you do? [a]
A. Right-click on an icon and look for it there or check View-Toolbars
B. Start a different module of Catia and look for it there
C. Close and start Catia again
D. Close the open document and open it again

4. Basic Catia V5: An assembly in Catia V5 has which suffix? [d]
A. *.Model B. *.CATPart C. *.Prt D. *.CATProduct

5. Part Design: When a new plane is created, where, in the specification tree, will it appear? (It is NOT a hybrid model) [a]
A. In the Geometrical Set.1 body B. In the Part Body
C. Just after the zx plane

6. Part Design: What is a draft feature? [b]
A. The function is often used when the part is going to be cast/forged/formed somehow with a tool
B. It is a feature you can use for creating lines
C. The function is often used within styling
D. You can create a pad with fillets with this command

7. Part Design: The term "parametric" means that a solid geometry is controlled by driving dimensions.
A. True [b]
B. False

8. Part Design: When is it a good idea to use multi-body techniques? [A]
A. For instance when designing complex and/or molded parts
B. When assemble complex products
C. When you have many surfaces
D. When you need to create a drawing

9. Part Design: What is a skeleton? [A]
A. Help geometry that you can use as reference
B.A thin walled CATIA model
C. Skeleton is another name for a sketch
D. Skeleton is another name for a geometrical set.

10. Part Design: What is true about sketch-based features? [B]
A. You need at least one sketch to create them
B. They can only be created in the Part Body (the first body)
C. It is an isolated sketch
D. They need a surface as parent element

11. Sketcher: When work is finished, why is it a good idea to use Sketch analysis? [C]
A. To verify that geometry in sketch is correct
B. To measure geometry in sketch
C. To change geometry in sketch
D. To change color/thickness of geometry in sketch

12. Sketcher: The Sketcher workbench is a standalone workbench that allows you to save a *.CATSketch file [B]
A. True B. False

13. Drafting: How do you create your first view (Front view)? [A]
A. Pick front view icon and pick a plane in the 3D viewer, Part or Assembly
B. Copy the object from assembly or part and paste it on the drawing
C. Pick a plane on object and right click choose in menu Front view
D. Drag the part or assembly into the drawing sheet

14. Drafting: How do you see which drawing view is active? [D]
A. The active view has a red frame
B. All views are active!
C. Only the active view is visible

15. Drafting: When you create an exploded view, you usually use a tool called Enhanced Scene. Why?

A. You can create an exploded view which doesn't affect the "master" product [a]
B. You can create position balloons there
C. You can create a Bill of material list there
D. You can create an animated movie there

16. Drafting: In a detailed drawing, are the views dependant on the CATParts point of origin?

A. True
B. False [a]

17. Assembly Design: When analyzing a component, Catia says it has 6 "degrees of freedom". What is the meaning of that? [a

A. It means that the component is fully loose in space
B. It means that the component is fully fixed in space
C. It means that the component is not saved
D. It means that the component is drafted with an angle of 6-degrees

18. Assembly Design: What is the name of the command that makes it possible to create an exploded view? [b]

A. Enhanced Scene B. 3D split C. Move D. Assembly split

19. Surface Design: When shall you use the Join command? [d

A. When you want to join two or more surfaces or curves
B. When you want to join a solid body to a surface
C. When I need to connect a sketch to a surface
D. When I want to join a point to a curve

20. Surface Design: What is an isolated element? [c]

A. Element without a history, it has no relation to a parent element
B. Element with a fixed color
C. Element you only use in Part Design
D. Element with a specific history, you can only change the length parameter

K.S.R.M. COLLEGE OF ENGINEERING (AUTONOMOUS), KADAPA-516003
DEPARTMENT OF MECHANICAL ENGINEERING
VALUE ADDED /CERTIFICATE COURSE ON
CATIA V5 R21 FROM 30/05/2022 TO 16/06/2022
ASSESSMENT TEST

12

Roll Number: 199Y1A0354 **Name of the Student:** S. Aslam

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

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

1. Basic Catia V5: A part in Catia V5 has which suffix? [b]
A. *.CATPart B. *.Model C. *.Prt D. *.Dwg

2. Basic Catia V5: Is "wrench-Bonn?et_string.CATPrpduct" an acceptable filename in Catia?
A. True B. False [b]

3. Basic Catia V5: If you are missing a toolbar, what do you do? [a]
A. Right-click on an icon and look for it there or check View-Toolbars
B. Start a different module of Catia and look for it there
C. Close and start Catia again
D. Close the open document and open it again

4. Basic Catia V5: An assembly in Catia V5 has which suffix? [d]
A. *.Model B. *.CATPart C. *.Prt D. *.CATProduct

5. Part Design: When a new plane is created, where, in the specification tree, will it appear? (It is NOT a hybrid model) [a]
A. In the Geometrical Set.1 body B. In the Part Body
C. Just after the zx plane

6. Part Design: What is a draft feature? [b]
A. The function is often used when the part is going to be cast/forged/formed somehow with a tool
B. It is a feature you can use for creating lines
C. The function is often used within styling
D. You can create a pad with fillets with this command

7. Part Design: The term "parametric" means that a solid geometry is controlled by driving dimensions.
A. True [a]
B. False

8. Part Design: When is it a good idea to use multi-body techniques? [a]
A. For instance when designing complex and/or molded parts
B. When assemble complex products
C. When you have many surfaces
D. When you need to create a drawing

9. Part Design: What is a skeleton? [a]
A. Help geometry that you can use as reference
B.A thin walled CATIA model
C. Skeleton is another name for a sketch
D. Skeleton is another name for a geometrical set.

10. Part Design: What is true about sketch-based features? [a]
A. You need at least one sketch to create them
B. They can only be created in the Part Body (the first body)
C. It is an isolated sketch
D. They need a surface as parent element

11. Sketcher: When work is finished, why is it a good idea to use Sketch analysis? [a]
A. To verify that geometry in sketch is correct
B. To measure geometry in sketch
C. To change geometry in sketch
D. To change color/thickness of geometry in sketch

12. Sketcher: The Sketcher workbench is a standalone workbench that allows you to save a *.CATSketch file [b]
A. True B. False

13. Drafting: How do you create your first view (Front view)? [b]
A. Pick front view icon and pick a plane in the 3D viewer, Part or Assembly
B. Copy the object from assembly or part and paste it on the drawing
C. Pick a plane on object and right click choose in menu Front view
D. Drag the part or assembly into the drawing sheet

14. Drafting: How do you see which drawing view is active? [C.]
A. The active view has a red frame
B. All views are active!
C. Only the active view is visible

15. Drafting: When you create an exploded view, you usually use a tool called Enhanced Scene. Why?

A. You can create an exploded view which doesn't affect the "master" product [a]

B. You can create position balloons there

C. You can create a Bill of material list there

D. You can create an animated movie there

16. Drafting: In a detailed drawing, are the views dependant on the CATParts point of origin?

A. True

B. False [a]

17. Assembly Design: When analyzing a component, Catia says it has 6 "degrees of freedom". What is the meaning of that? [C

A. It means that the component is fully loose in space

B. It means that the component is fully fixed in space

C. It means that the component is not saved

D. It means that the component is drafted with an angle of 6-degrees

18. Assembly Design: What is the name of the command that makes it possible to create an exploded view? [C]

A. Enhanced Scene B. 3D split C. Move D. Assembly split

19. Surface Design: When shall you use the Join command? [b

A. When you want to join two or more surfaces or curves

B. When you want to join a solid body to a surface

C. When I need to connect a sketch to a surface

D. When I want to join a point to a curve

20. Surface Design: What is an isolated element? [C]

A. Element without a history, it has no relation to a parent element

B. Element with a fixed color

C. Element you only use in Part Design

D. Element with a specific history, you can only change the length parameter

Certification Course
on
CATIA V5 R21

Notes of

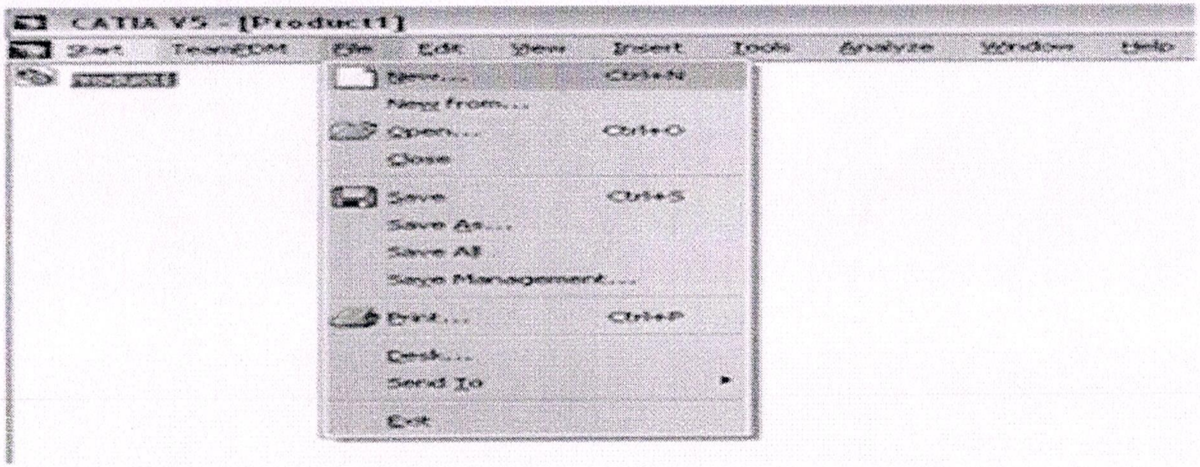
CATIA V5 – CAD

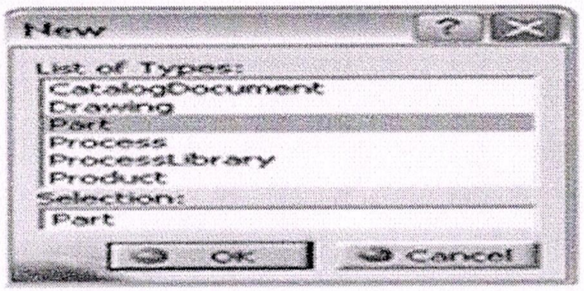
-: INDEX :-

Entering the Software Scenario:

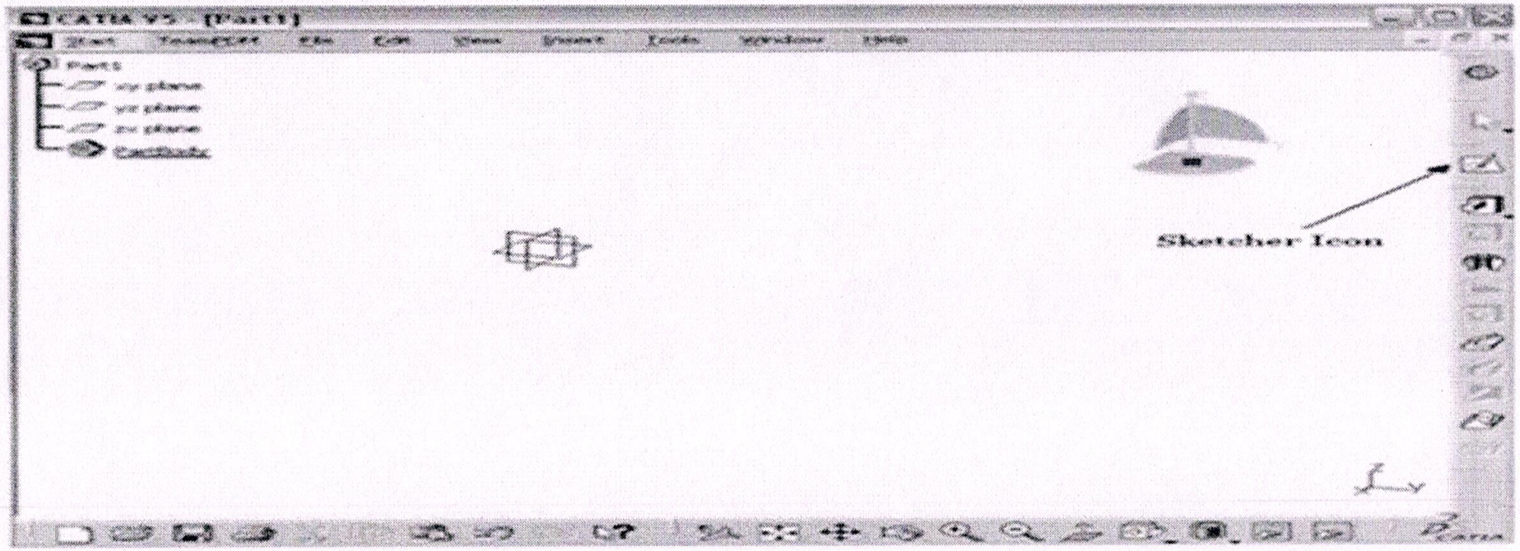
By Default the CATIA opens in CATIA Product Application. To go in CATIA Part Design Application.

Click File->New->Part.





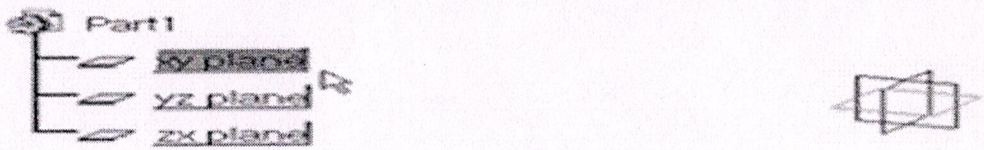
Click **OK** to enter in to Part design workbench.
This is how Part Design Workbench looks



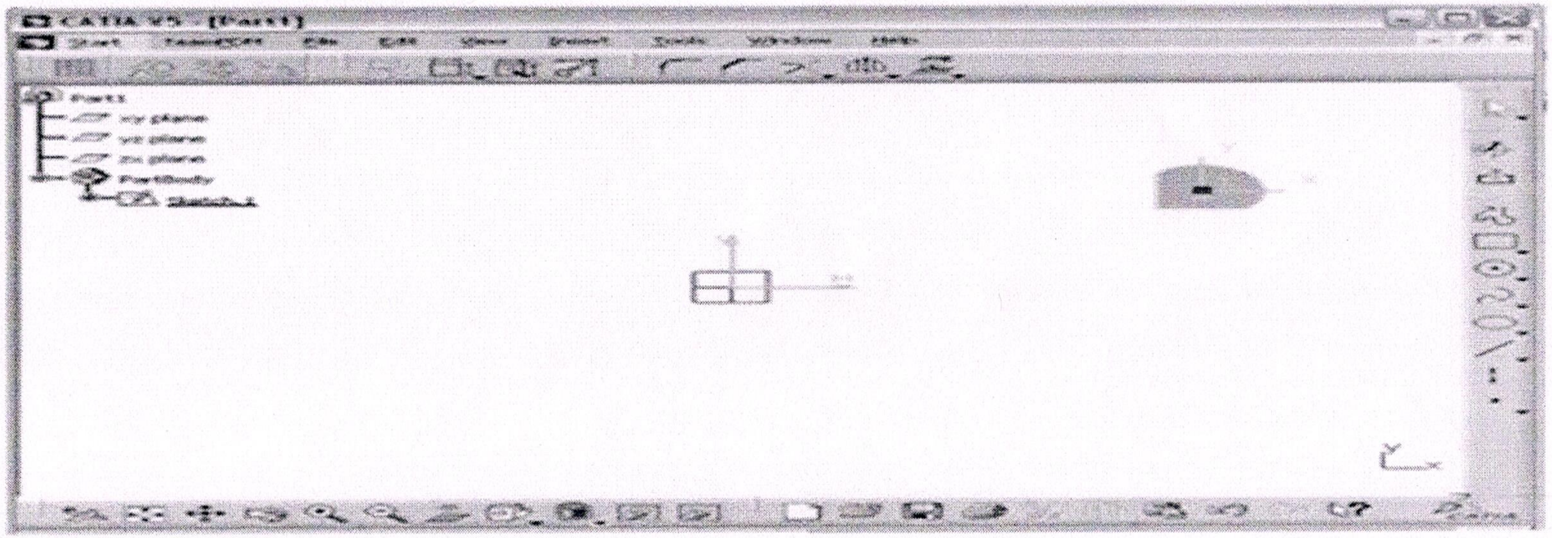
Entering the Sketcher Workbench:

This task lists the different ways of entering the Sketcher workbench. To create a sketch, you have two possibilities,

- Select **Start -> Mechanical Design -> Sketcher** from the menu bar.
- Select the **Sketcher** icon  and click the desired reference plane either in the geometry area or in the specification tree, or select a planar surface.



This is how Sketcher Workbench looks.



Sketcher:

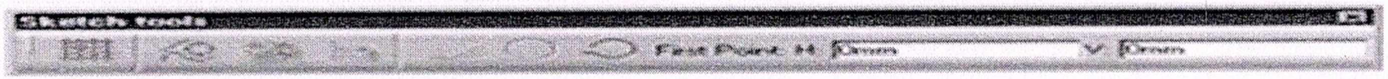
The following sketcher icons are offered by this workbench.

Profiles:

Profile: 

This task shows how to create a closed profile. A profile may also be open (if you click the profile end point in the free space). Profiles may be composed of lines and arcs which you create either by clicking or using the Sketch tools toolbar.

The Sketch tools toolbar appears with option commands and values:

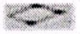


Line (active by default)  Tangent Arc  Three Point Arc

Predefined profiles:

Rectangle: 

Create rectangle by using two diagonal points.

Oriented rectangle: 

Create oriented rectangle by using three points.

Parallelogram: 

Create parallelogram by selecting three points.

Elongated hole: 

First two points will define center to center (CTC) distance; third point pt will define the radius.

Cylindrical elongated hole: 

First pt will define the circle, second and third will define the arc and last pt will define the radius of hole.

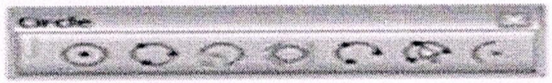
Key hole:

First two  pts will define the CTC distance, third and fourth would define two radii.

Hexagon:

First pt  would define the center of hexagon, second would define the center point of side of hexagon.

Circle:



Circle: 

Circle using two points one for center and one for radius.

Three point circle: 

Circle is created through three points.

Circle by co-ordinates: 

Circles created by entering the center pt coordinates and radius value.

Tri-tangent circle: 

By selecting three elements, a tangential circle is created.

Arc by three points: 

An arc created through three points.

Arc by limit: 

Arc is created by first pt as start pt, second as end pt and third pt anywhere on arc.

Arc: 

Arc is created by center pt and radius.

Spline: 

Indicate the points through which the spline goes. Double-click the last point you have created to finish the spline creation.

Ellipse: 

An ellipse is created by center point, major and then minor semi-axis endpoint.

Line:

Lines: 

A line is created between two points

Infinite line: 

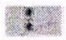
An infinite line is created on screen.

Bi-tangent line: 

A tangential line created to two elements.

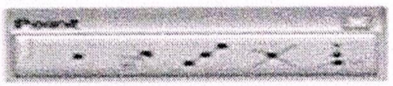
Bisecting lines: 

A bisecting line is created between two selected lines.

Axis: 

This task shows how to create an axis. You will need axis whenever creating shafts and grooves. Axis cannot be converted into construction elements.

Here also you have to click first for star point and then second to end point, an axis is created between those two points.



Points: 

A point is created by clicking on screen.

Points by co-ordinates: 

A point is created by entering the co-ordinate in H and V directions.

Equidistance points: 

Equidistance points are created on selected element by entering the numbers.

Intersection point: 

A point is created at intersection of two elements.

Projection point:

A projected point is created on selected element.

Constraints:

There are times when simple sketches are adequate for your design process, but you will often need to work on more complex sketches requiring a rich set of geometrical or dimensional constraints. The Sketcher workbench provides constraint commands which will allow you to fully sketch your profiles.

Create Quick Dimensional/Geometrical Constraints: Set constraints on elements or between two or three elements. The constraints are in priority dimensional. Use the contextual menu to get other types of constraints and to position this constraint as desired.

Create Contact Constraints: Apply a constraint with a relative positioning that can be compared to contact. You can either select the geometry or the command first. Use the contextual menu if you want to insert constraints that are not those created in priority.

Create Constraints Using a Dialog Box: Set various geometrical constraints between one or more elements using a dialog box and if needed, multi-selection.

Auto-Constrain a Group of Elements: Detects possible constraints between selected elements and imposes these constraints once detected.

Animate Constraints: Assign a set of values to the same angular constraint and examine how the whole system is affected.

Quickly Creating Dimensional/Geometrical Constraints:

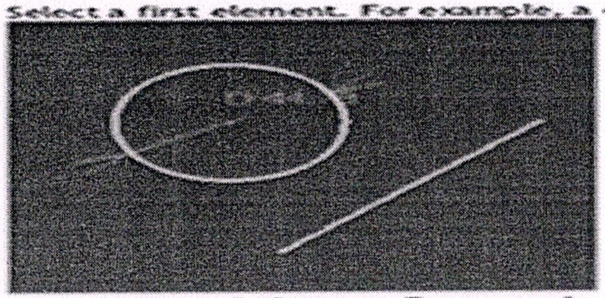
This task shows you how to set dimensional or geometrical constraints between one, two or three elements. The constraints are in priority dimensional.

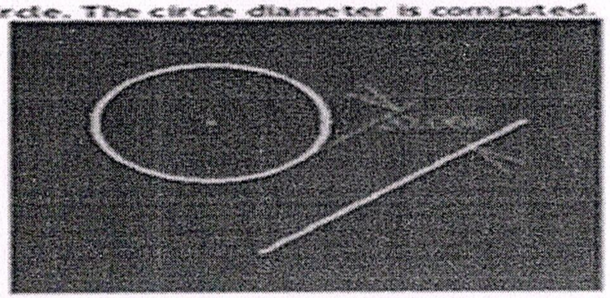
In this particular case, we will set constraints between two elements by selecting the command and then a line and a circle.

But what you can also do is set dimensional constraints by multi-selecting the circle and line, and then clicking the **Constraint** icon. At any time, you may move the cursor: the distance value will vary accordingly. Click for positioning the newly created dimensional constraint.

Select the **Constraint** icon from the **Constraint** toolbar.

Select a first element. For example, a circle. The circle diameter is computed.





Select a second element. For example, a line.

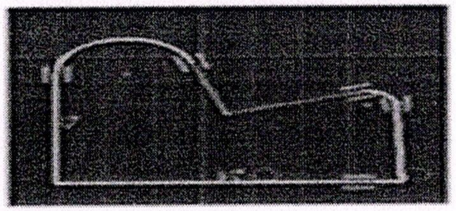
The relation between the two elements is reconsidered. In other words, the diameter of the circle is no longer displayed.

Creating Constraints via a Dialog Box:

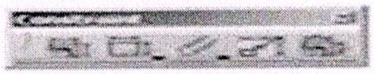
This task shows you how to set various geometrical constraints using a dialog box. For example, you can use the Constraint command to finalize your profile and set constraints consecutively.

You may define several constraints simultaneously using the **Constraint Definition** dialog box.

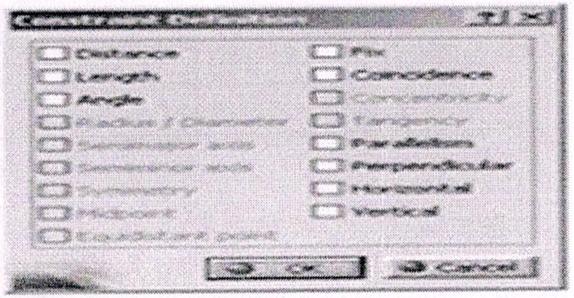
Multi-select the elements to be constrained. For example, two lines.



Click the **Constraints Defined in Dialog Box** icon  from the **Constraint** toolbar.

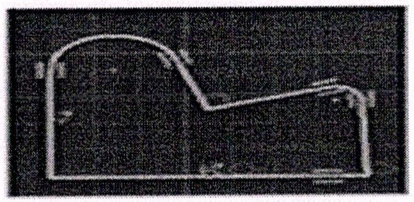


The **Constraint Definition** dialog box appears, indicating the types of constraints you can set between the selected lines (selectable options).



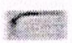
- These constraints may be constraints to be applied either **one per element (Length, Fix, Horizontal, and Vertical)** or constraints **between two selected elements (Distance, Angle, Coincidence, Parallelism or Perpendicular)**.
- Multi-selection is available.
- If constraints already exist, they are checked in the dialog box, by default.

Check the **Perpendicular** option to specify that you want the lines to always remain perpendicular to each others, whatever ulterior modifications. Click **OK**.

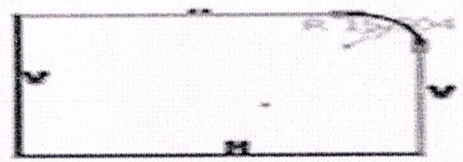


The perpendicularity symbol appears.

Operations:

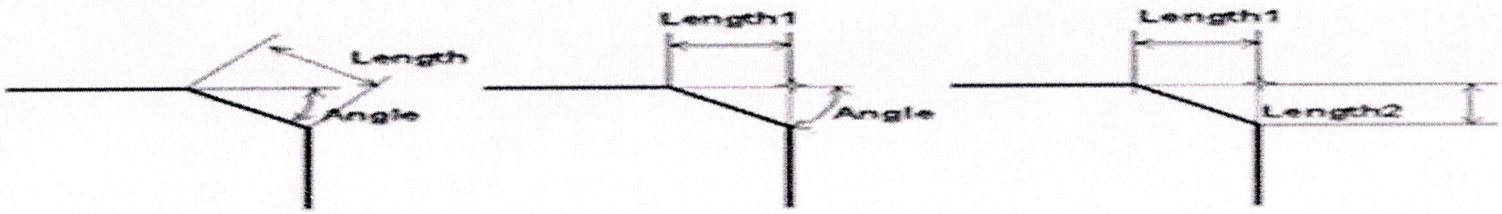
Creating Corners: 

Select the first line. The selected line is highlighted. Select the second line. The second line is also highlighted, and the two lines are joined by the rounded corner which moves as you move the cursor. This lets you vary the dimensions of the corner.



Creating Chamfers:

This task shows how to create a chamfer between two lines trimming either all, the first or none of the elements, and more precisely using one of the following chamfer definitions:

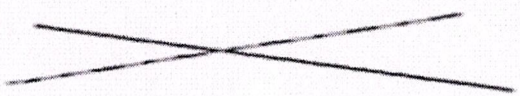


Relimitation operations:

Trimming Elements:

This task shows how to trim two lines (either one element or all the elements).

Select the first line. The selected element is highlighted.



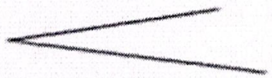
Position the cursor on the element to be trimmed.

The second element is highlighted too, and both lines are trimmed.

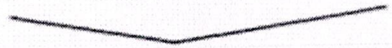
If you position the cursor on the same first element, it will be trimmed at the location of the second position.

The location of the relimitation depends on the location of the cursor.

Click when you are satisfied with the relimitation of the two lines.



First example



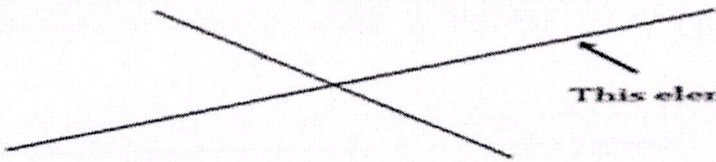
Second example

Trimming:

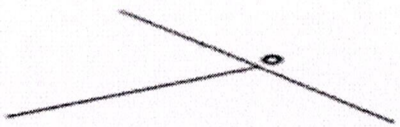
This task shows how to quickly delete elements intersected by other Sketcher elements using breaking and trimming operations.

Click the **Quick Trim** icon from the **Operation** toolbar (**Relimitations** sub toolbar).

Select part of element which you want to trim.



Then result will be...

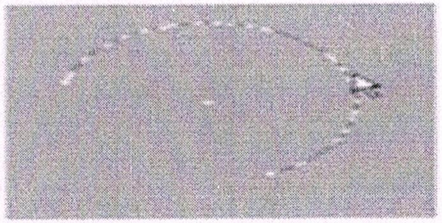


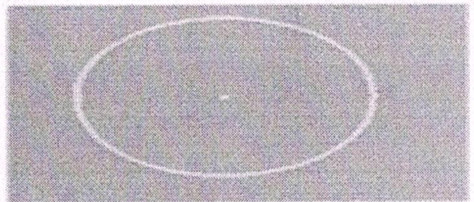
Breaking Elements:

This task shows how to break a line using a point on the line and then a point that does not belong to the line. The **Break** command lets you break any type of curve, except composite curves. You can use any Sketcher element to break curves.

Closing Elements:

Click the **Close** icon from the **Operation** toolbar (**Relimitations** sub toolbar).
Select one or more elements to be relimited. For example, a three point arc.

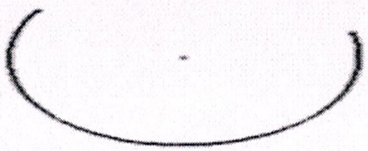




The arc is closed.

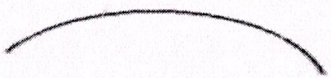
Complementing an Arc (Circle or Ellipse):

Click on the arc to be complemented to select it. For example, the three points arc.



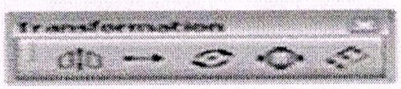
Click the **Complement** icon

Select the arc



The compliment of the arc is created.

<u>**Transformation operations:**</u>

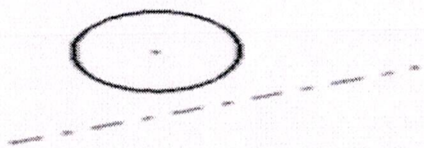


Creating symmetrical Elements:

This task shows you how to repeat existing Sketcher elements using a line, a construction line or an axis. In this particular case, we will duplicate a circle.

Create a circle and line.

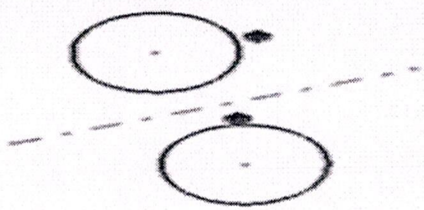
Select the circle to be duplicated by symmetry.



Click the **symmetry** icon from the **Operation** toolbar.

Select the axis you previously created.

The selected circle is duplicated and a symmetry constraint is created.



Translating Elements:

This task will show you how to perform a translation on 2D elements by defining the duplicate mode and then selecting the element to be duplicated. Multi-selection is not available.

You may either perform a simple translation (by moving elements) or create several copies of 2D elements.

Translating elements also means re-computing distance, angle and/or length constraint values, if needed. Be careful: only non-fixed elements are updated.

Rotating Elements:

This task will show you how to rotate elements by defining the duplicate mode and then selecting the element to be duplicated.

In this scenario, the geometry is simply moved. But note that, you can also duplicate elements with the Rotation command by giving center for rotation.

Rotating elements also means re-computing distance values into angle values, if needed. Be careful: only non-fixed elements are updated.

Scaling Elements: 

This task will show you how to scale an entire profile. In other words, you are going to resize a profile to the dimension you specify.

Here you to give a scaling factor and reference point to scale.

Scaling elements also means re-computing distance values, if needed. Note that angle values will not be modified. Be careful: only non-fixed elements are updated.

<u>3D geometry:</u>

Project 3d element: 

Projects 3D Elements onto the Sketch Plane.

Intersecting 3d element: 

Intersects 3D Elements with the Sketch Plane.

Project3d silhouette edge: 

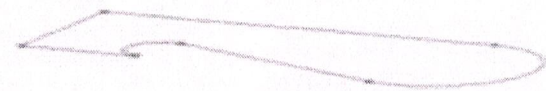
Projects 3D Silhouette Edges.

Entering the Part Mode:

Pad:

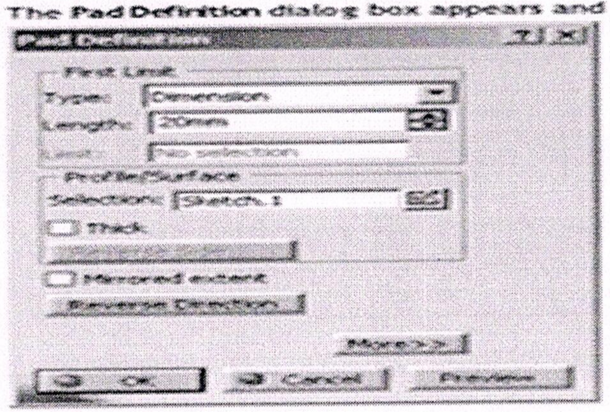
Creating a pad means extruding a profile or a surface in one or two directions. The application lets you choose the limits of creation as well as the direction of extrusion.

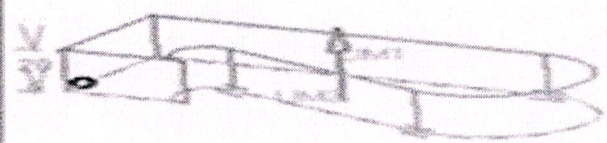
Select **Sketch** as the profile to be extruded.



Click the **Pad** icon 

The **Pad Definition** dialog box appears and the application previews the pad to be created.





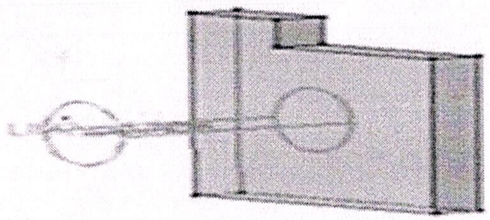
Limits:

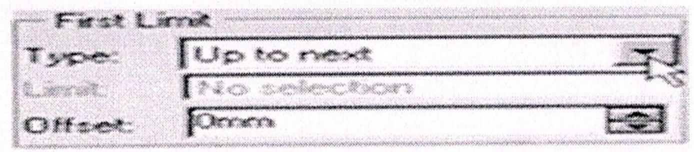
You will notice that by default, the application specifies the length of your pad (Type= Dimension option). But you can use the following options too:

- **UP TO NEXT**
- **UP TO LAST**
- **UP TO PLANE**
- **UP TO SURFACE**

Up to Next:

In the **Type** field, set the option to **Up to next**



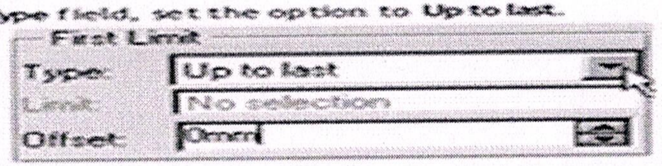


This option assumes an existing face can be used to trim the pad. The application previews the pad to be created. The already existing body trims the extrusion.

Optionally, click **Preview** to see the result.

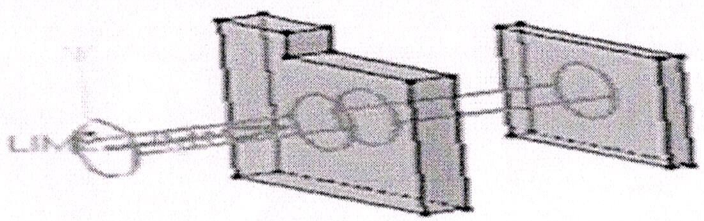
Up to Last:

In the **Type** field, set the option to **Up to last**.



The last face encountered by the extrusion trims the pad.

Optionally, click **Preview** to see the result.



Up to Plane:

In the **Type** field, set the **Type** option to **Up to plane**

Select **Plane** The application previews the pad to be created. The plane trims the extrusion. An **Offset** option is now displayed. Enter the offset value. This offset is the distance between the plane and the top face of the pad to be created.

Up to Surface:

In the **Type** field, set the **Type** option to **Up to surface**

Select the surface or face up to which you want to pad the sketch. This face belongs to the same body as the existing pad.

Here also the offset option is displayed, this also operates in the same manner as up to plane option.

Multi-Pad:

Multi-Pad icon

Select **Sketch** that contains the profiles to be extruded. Note that all profiles must be closed and must not intersect. In case a profile would be open, the application would not take it into account.

The **Multi-Pad Definition** dialog box appears and the profiles are highlighted in green. For each of them, you can drag associated manipulators to define the extrusion value.

The **Multi- Pad Definition** dialog box displays the number of domains to be extruded.
Selected **Extrusion domain** appears in blue in the geometry area.
Specify the length by entering a value. You need to repeat the operation for each extrusion domain by entering the Value.

Drafted Filleted Pad:

Drafted Filleted Pad icon

This task shows you how to create a pad while drafting its faces and filleting its edges.

Pocket:

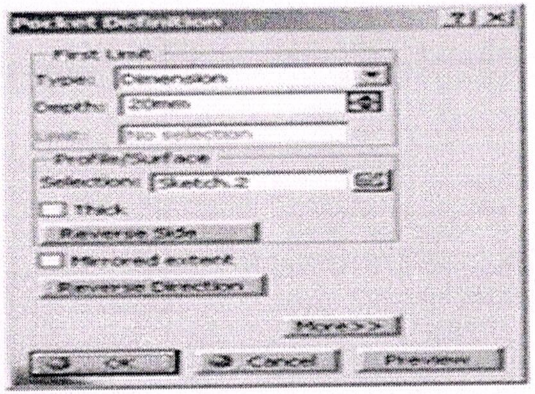
Creating a pocket consists in extruding a profile or a surface and removing the material resulting from the extrusion. The application lets you choose the limits of creation as well as the direction of extrusion. The limits you can use are the same as those available for creating pads. To know how to use them, see Up to Next Pads , Up to Last Pads, Up to Plane Pads, Up to Surface Pads.

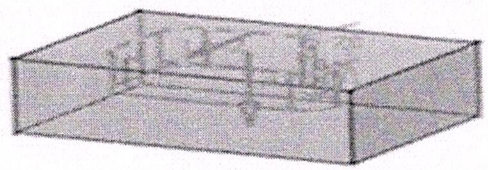
This task first shows you how to create a pocket, which is a cavity, in an already existing part.

Select the profile to extrude that is **Sketch.**

Click the **Pocket** icon.

The **Pocket Definition** dialog box is displayed and the application previews a pocket.

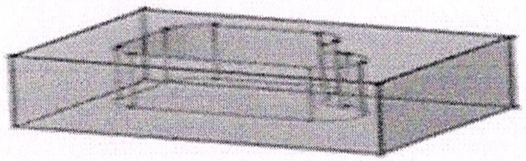




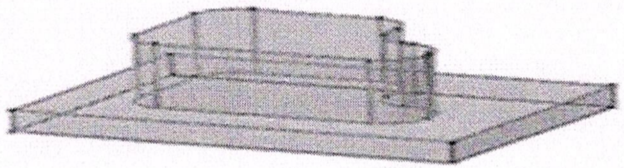
You can define a specific depth for your pocket or set one of these options:

- **UP TO NEXT**
- **UP TO LAST**
- **UP TO PLANE**
- **UP TO SURFACE**

click **Preview** to see the result. Click **OK** to create the pocket.



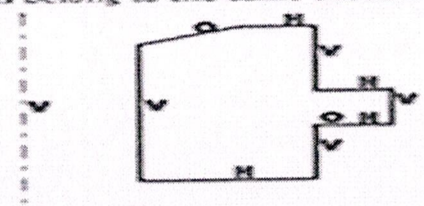
The **Reverse side** option lets you choose between removing the material defined within the profile, which is the application's default behavior, or the material surrounding the profile.



Shaft:

This task illustrates how to create a shaft that is a revolved feature, by using a profile.

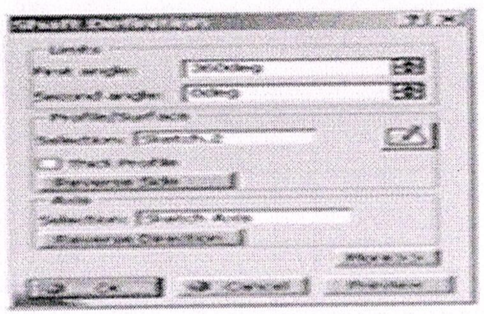
Select **Sketch** as the profile to be extruded. For the purposes of our scenario, the profile and the axis belong to the same sketch.



Exit the workbench.

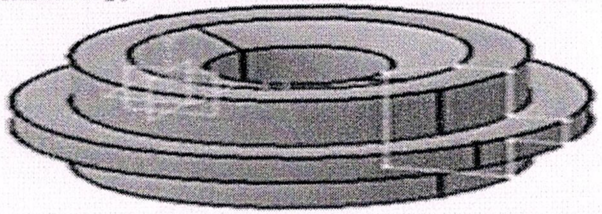
Click the **Shaft** icon 

The **Shaft Definition** dialog box is displayed



If you have already drawn axis in sketch you need not to select it again ,but if you want have axis other than that, then either you need to create it in another sketch or you have to select existing part's cylindrical face so as to mach with that solid.

Enter the values of your choice in the fields **First angle** and **Second angle**.
Alternatively, select the **LIM1** or **LIM2** manipulator and drag them onto the value of your choice.



Click **OK** to confirm.

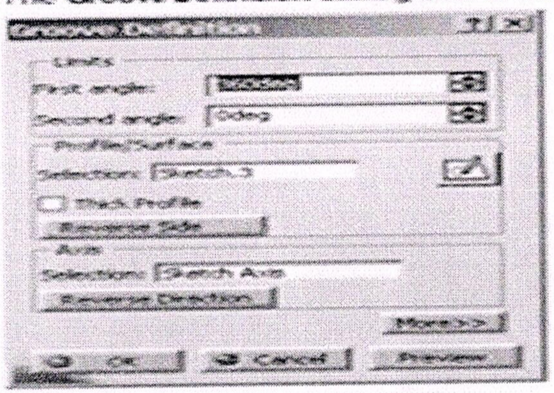
Groove:

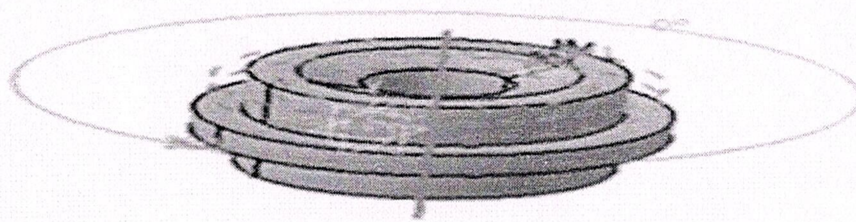
Grooves are revolved features that remove material from existing features. This task shows you how to create a groove that is how to revolve a profile about an axis (or construction line).

Click the **Groove** icon

Select **Sketch** as the profile to be used.

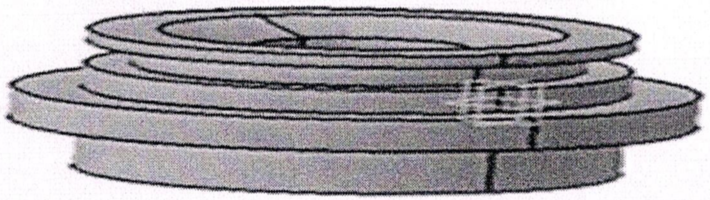
The **Groove Definition** dialog box is displayed





Select any cylindrical surface for axis.

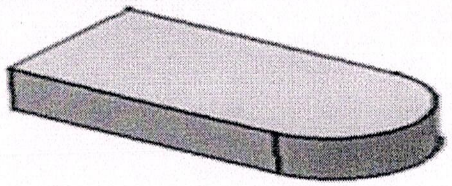
Click **OK** to confirm the operation



Holes:

This task shows how to constrain the location of the hole to be created without using the Sketcher workbench's tools.

Multi-select both edges as shown and the upper face which is the face on which you wish to position the hole.



Click the **Hole** icon

The preview displays two constraints defining the distances between the hole's center and the edges.

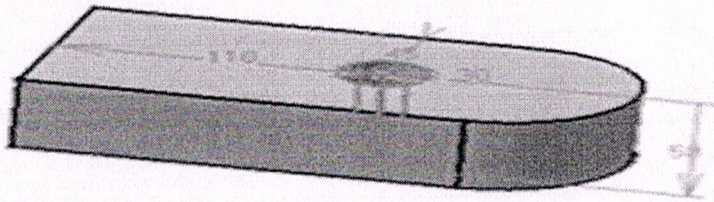
Define the parameters in the dialog box to create the desired hole

The application previews the constraints you are creating.

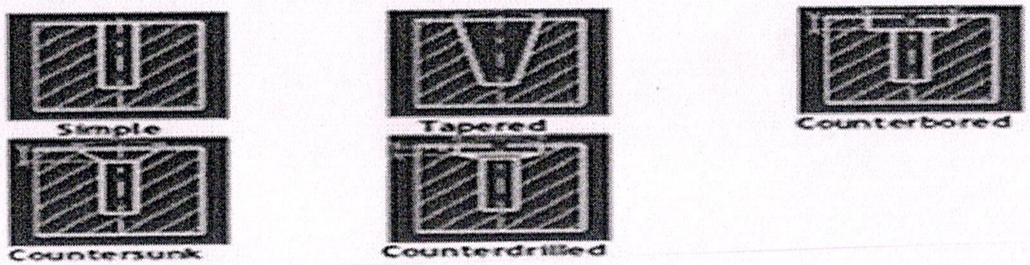
To access the constraint values, double-click the constraint of interest. This displays the **Constraint Definition** dialog box in which you can edit the value.

Click **OK** to create the hole.

The application positions the hole using constraints.



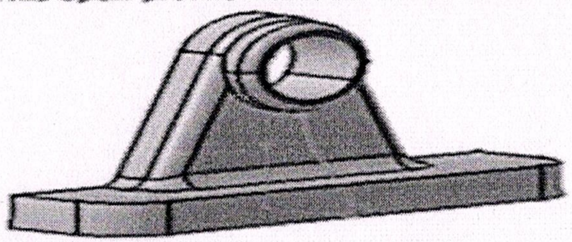
There are many other options by which you can create the following types of Holes.



Stiffener:

This task shows you how to create a stiffener by specifying creation directions
Select the profile to be extruded that is **Sketch** (located in the Part Body entity).

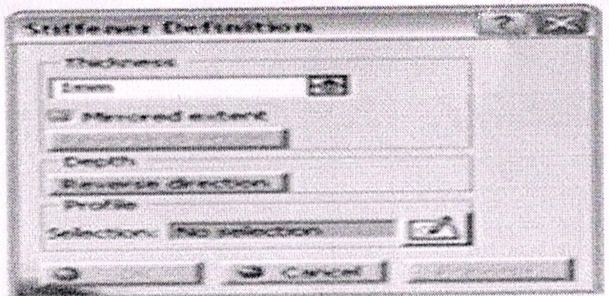
This open profile has been created in a plane normal to the face on which the stiffener will lie.



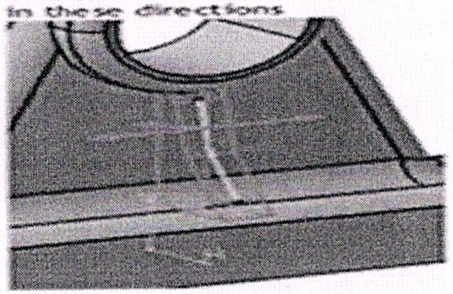
If you need to use an open profile, make sure that existing material can fully limit the extrusion of this profile

Click the **Stiffener** icon 

The **Stiffener Definition** dialog box is displayed.

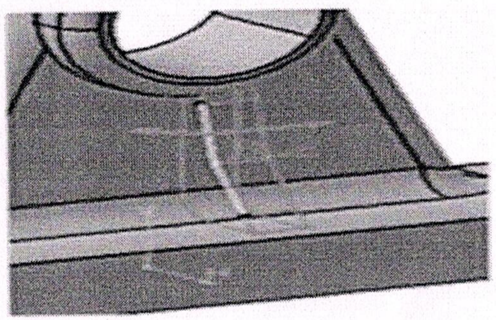


The extrusion will be made in three directions, two of which are opposite directions. Arrows point in these directions



Uncheck the **Neutral Fiber** option.

The extrusion will be made in two directions only. To obtain the directions you need, you can also click the arrows



Check the **Neutral Fiber** option again.

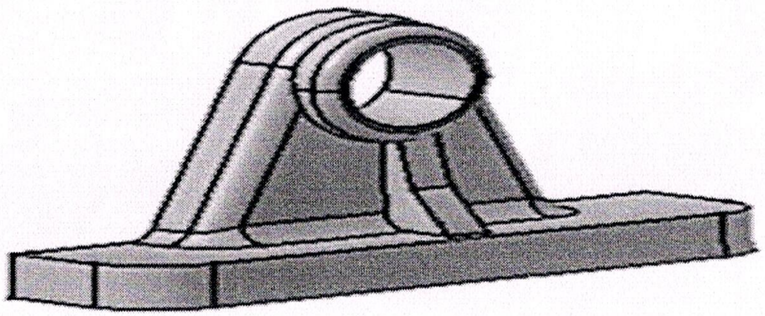
This option adds material equally to both sides of the profile.

Enter x as the thickness value.

This thickness is now evenly distributed: a thickness of x/2 is added to each side of the profile.

Optionally click **Preview** to see the result.

Click **OK**.



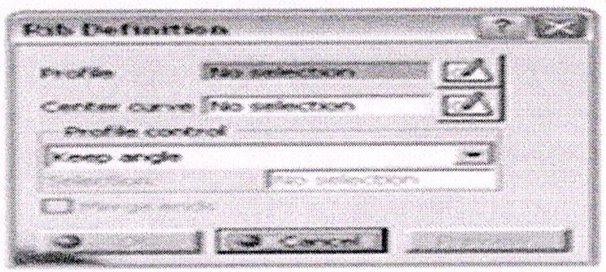
Ribs and slots:

This task shows you how to create a rib that is how to sweep a profile along a center curve to create material.

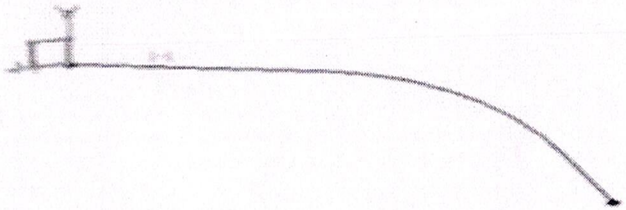
To define a rib, you need a center curve, a planar profile and possibly a reference element or a pulling direction. You can combine the different elements as follows:

Click the **Rib** icon. 

The **Rib Definition** dialog box is displayed



Select the profile you wish to sweep, i.e. Your profile has been designed in a plane normal to the curve used to define the center curve. It is a **closed profile**.



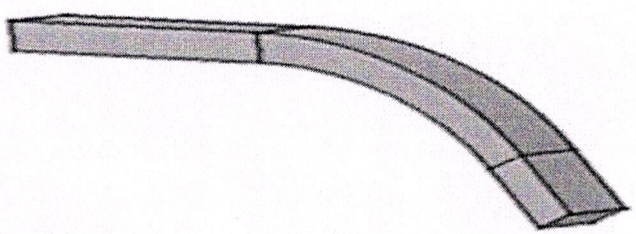
Profile Control

You can control its position by choosing one of the following options:

- **Keep angle**: keeps the angle value between the sketch plane used for the profile and the tangent of the center curve.
- **Pulling direction**: sweeps the profile with respect to a specified direction. To define this direction, you can select a plane or an edge. For example, you need to use this option if your center curve is a helix. In this case, you will select the helix axis as the pulling direction.
- **Reference surface**: the angle value between axis h and the reference surface is constant.

To go on with our scenario, let's maintain the **Keep angle** option. Remember, the angle value is 90 degrees.

Click **OK**.



The rib is created.

Slot:

This task shows you how to create a slot that is, how to sweep a profile along a center curve to remove material. This command is same as the rib the only difference is it removes the material.

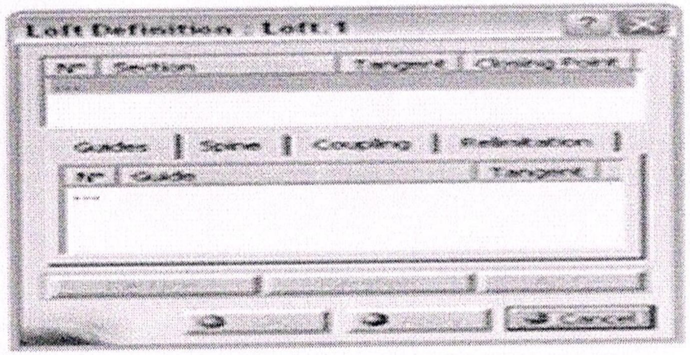
Loft:

This task shows how to create a multi-sections solid.

You can generate it by sweeping one or more planar section curves along a computed or user-defined spine. The feature can be made to respect one or more guide curves. The resulting feature is a closed volume.

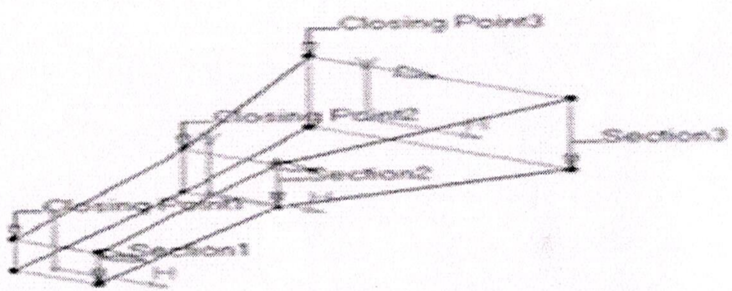
Click the **Loft** icon 

The **Loft Definition** dialog box appears.



Select the three section curves as shown:

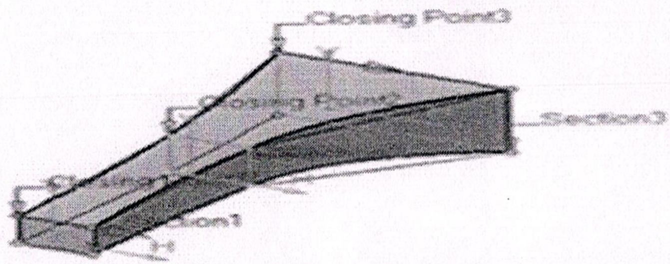
They are highlighted in the geometry area.



The Loft capability assumes that the section curves to be used do not intersect.

Click **Preview** to preview the feature to be created.

You can note that by default, tangency discontinuity points are coupled



Several coupling types are available in the **Coupling** tab:

- **Ratio**: the curves are coupled according to the curvilinear abscissa ratio.
- **Tangency**: the curves are coupled according to their tangency discontinuity points. If they do not have the same number of points, they cannot be coupled using this option.
- **Tangency then curvature**: the curves are coupled according to their curvature discontinuity points. If they do not have the same number of points, they cannot be coupled using this option.
- **Vertices**: the curves are coupled according to their vertices. If they do not have the same number of vertices, they cannot be coupled using this option

For the purpose of our scenario, you are going to use guide curves. Click the Guide field and select the four joins. The curves to be used must be joined.

They are highlighted in the geometry area.

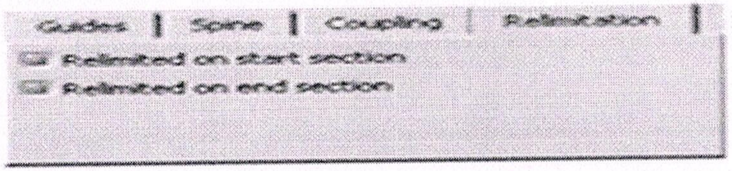
It is possible to edit the Loft reference elements by first selecting a curve in the dialog box list then choosing a button to either:

- Remove the selected curve
- Replace the selected curve by another curve.
- Add another curve.

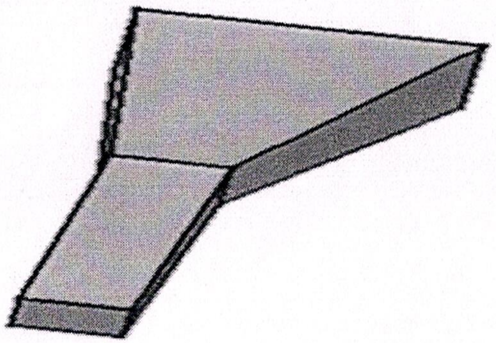
By default, the application computes a spine, but if you wish to impose a curve as the spine to be used, you just need to click the **Spine** tab then the **Spine** field and select the spine of your choice in the geometry.

The **Relimitation** tab lets you specify the feature relimitation type. You can choose to limit the Loft only on the Start section, only on the End section, on both, or on none.

- when one or both are checked: the Loft is limited to corresponding section
- when one or both are when unchecked: the Loft is swept along the spine:
- if the spine is a user spine, the Loft is limited by the spine extremities
- if the spine is an automatically computed spine, and no guide is selected: the feature is limited by the start and end sections
- if the spine is an automatically computed spine, and guides are selected, the feature is limited by the guides extremities.



Click **OK** to create the volume.



Removed loft:

Removed Loft icon

This task shows how to remove a multi-sections solid.

The **Loft** capability generates lofted material surface by sweeping one or several planar section curves along a computed or user-defined spine then removes this material. The material can be made to respect one or more guide curves.
All other functions are same as the Loft.

Fillet:

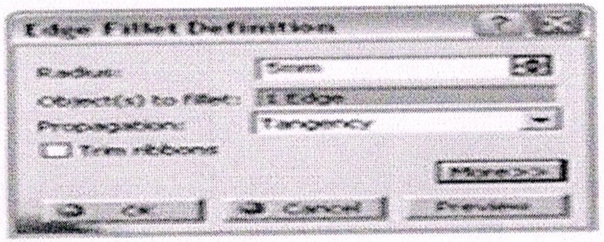
Edge Fillet:

A fillet is a curved face of a constant or variable radius that is tangent to, and that joins, two surfaces. Together, these three surfaces form either an inside corner or an outside corner.

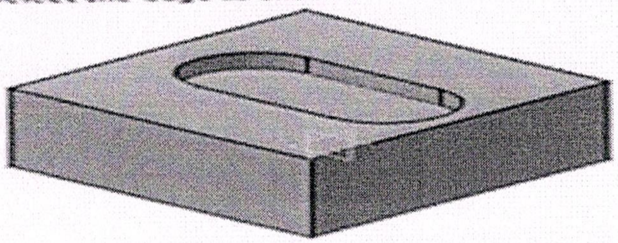
In drafting terminology, the curved surface of an outside corner is generally called a round and that of an inside corner is normally referred to as a fillet

Click the **Edge Fillet** icon

The **Edge Fillet Definition** dialog box appears.



Select the edge as shown.

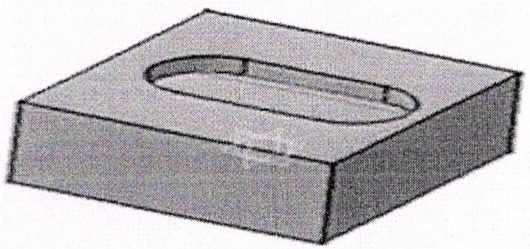


The edge selected then appears in the **Objects** to fillet field. The application displays the radius value.

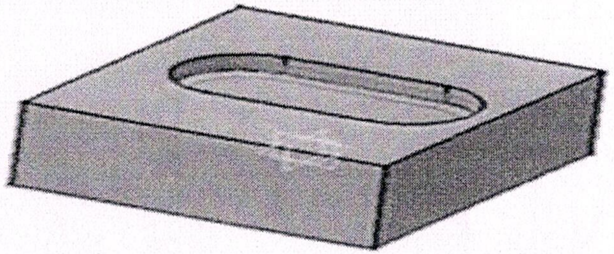
Clicking **Preview** previews the fillet to be created.

Two propagation modes are available:

- **Minimal**: edges tangent to selected edges can be taken into account to some extent. The application continues filleting beyond the selected edge whenever it cannot do otherwise. In our example below, the fillet is computed on the selected edge and on a portion of tangent edges:

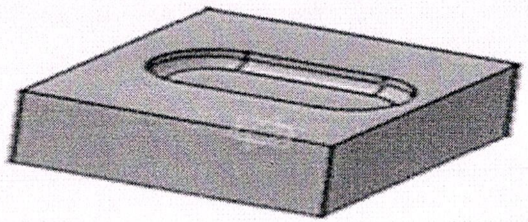


- **Tangency**: tangencies are taken into account so as to fillet the entire edge and possible tangent edges.



For the purpose of our scenario, set the **Tangency** option. The preview clearly shows that the whole edge will be filleted.

Enter radius value. The radius value is updated in the geometry area.



Click **OK**.

Tritangent Fillet:

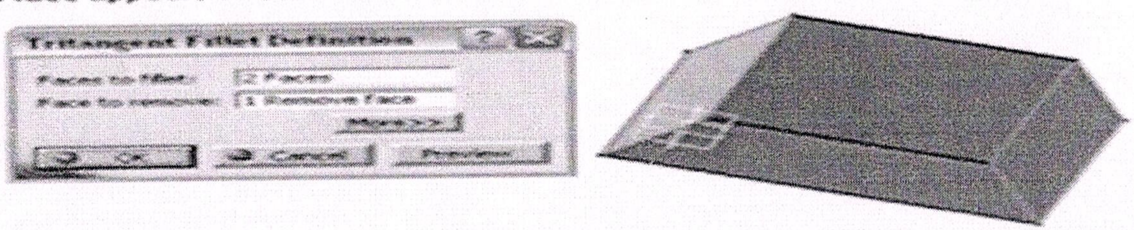
The creation of tritangent fillets involves the removal of one of the three faces selected

Click the **Tritangent Fillet** icon. 

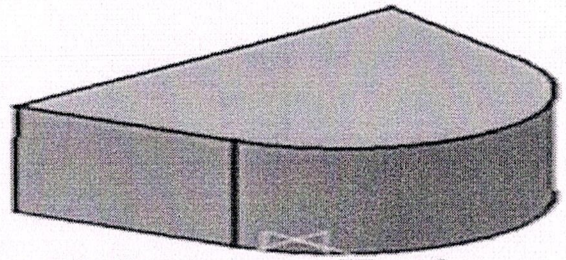
The **Tritangent Fillet Definition** dialog box appears.

Select the faces to be filleted.
Select the face to be removed, that is the upper face. The fillet will be tangent to this face.

This face appears in dark red.



Optionally, click **Preview** to see the fillet to be created.



Click **OK**. The faces get filleted.

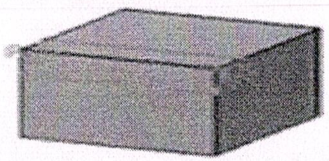
Variable Radius Fillet:

Variable radius fillets are curved surfaces defined according to a variable radius. A variable radius corner means that at least two different constant radii are applied to two entire edges.

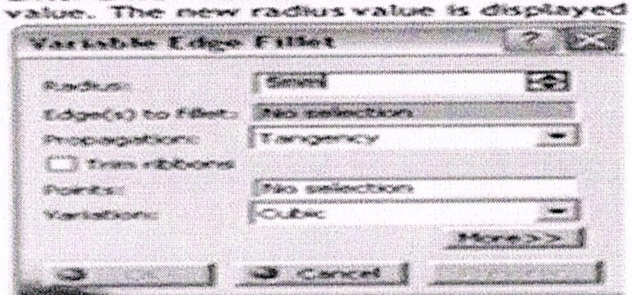
Click the **Variable Radius Fillet** icon.

The **Variable Radius Fillet** Definition dialog box appears.
Select the edge to be filleted. You can define variable radius fillets on closed edges.

The application detects both vertices and displays two identical radius values.



Enter a new radius value to simultaneously change the radius of both vertices. For example, enter value. The new radius value is displayed on both vertices. The preview is modified accordingly.

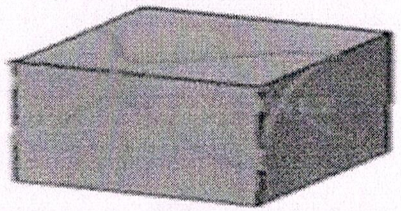


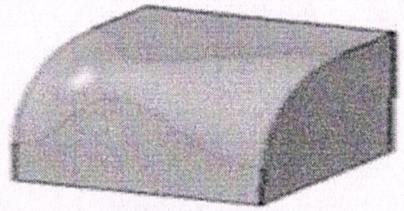
Enter different values at both vertices.

To add a point on the edge to make the variable radius fillet more complex, click the **Points** field. You can add points by selecting planes or edge.

Enter a new radius value for this point
The new radius value is displayed

The variation mode is set to **Cubic**: keep this mode.
In the **linear** option the joining will be point to point.





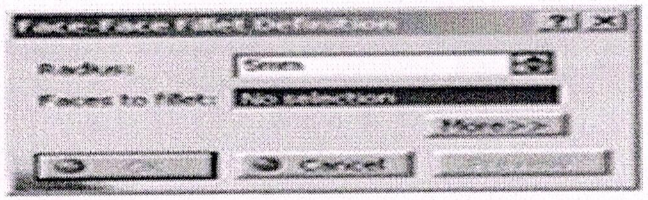
Click **OK** to confirm the operation.

The edge is filleted.

Face to Face Fillet:

You generally use the Face-face fillet command when there is no intersection between the faces or when there are more than two sharp edges between the faces.

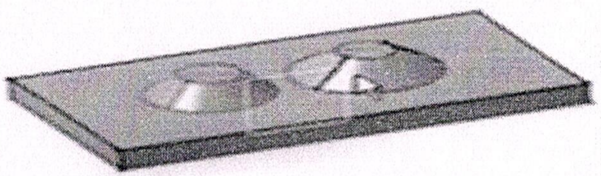
Click the **Face-Face Fillet** icon. 

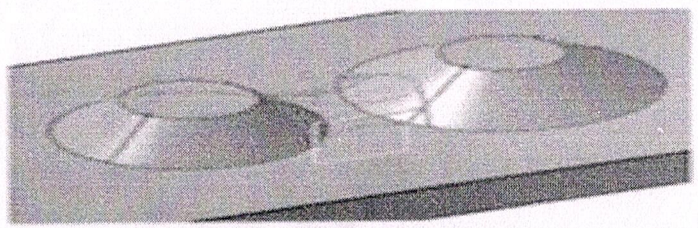


The **Face-Face Fillet** Definition dialog box appears.

Select the faces to be filleted. click **Preview** to see the fillet to be created.

Enter a radius value in the **Radius** field





Click **OK**.