

Experiment No. 1 – Spur Gear Design in CATIA V5R21 (Using Formulas)

Aim:

- To design a spur gear with 25 teeth and a module of 3.5 mm using CATIA V5R21 by applying standard gear calculations, formulas, and 3D modeling commands.

Hardware and Software Required:

✓ Software:

- CATIA V5R21 or higher
- Windows 10 or above

✓ Hardware:

- 8 GB RAM or more
- Intel i5/i7 or AMD Ryzen processor
- Graphics support for 3D CAD
- Mouse with scroll and middle button

Given:

- Number of Teeth, $N = 25$
- Module, $m = 3.5 \text{ mm}$
- Pressure Angle, $\alpha = 20^\circ$

Formula Used:

S. No	Parameter	Formula	Value (mm)
1	Pitch Circle Radius (R_p)	$R_p = (m \times N) / 2$	43.75
2	Clearance Circle Radius (R_b)	$R_c = 0.94 \times R_p$	41.125
3	Addendum Circle Radius (R_a)	$R_a = R_p + m$	47.25
4	Dedendum Circle Radius (R_d)	$R_d = R_p - 1.25 \times m$	39.375
5	Quarter of the Gear Index Angle	$(90/N) \times 1^\circ$	3.6°
6	Fillet Radius	$0.39 \times m$	1.365

CATIA Commands Used:

- Part Design
- Sketch
- Formula Editor

- Pad (Mirror Extend)
- Pocket (Mirror Extend)
- Circular Pattern (Complete Crown)
- Mirror
- Fillet

Procedure:

1. Create a New Part in Part Design and name it 'SpurGear_25T_3.5m'.
2. Open Formula Editor and define input parameters: Module = 3.5 mm, No of Teeth = 25.
3. Add gear formulas for Pitch Radius, Clearance Radius, Addendum Radius, Dedendum Radius, Clearance Radius, Fillet Radius, Gear Index Angle, and Quarter Angle using standard equations.
4. Create a Sketch on the XY plane and draw four concentric circles using the radii: R_p , R_a , R_d , and R_c .
5. Sketch $\frac{1}{4}$ Tooth Profile using radial lines and apply the fillet at the root; then mirror to complete one full tooth and trim excess lines.
6. Use the Pad Command with Mirror Extend to create the gear body, total thickness = 10 mm.
7. Create a New Sketch to define the gap between teeth, and use the Pocket Command with Mirror Extend (10 mm total depth).
8. Apply Circular Pattern in Complete Crown mode to duplicate the pocketed tooth around 360° , with 25 instances.
9. Add Fillets on gear edges if required to smoothen sharp corners.
10. Save the Final Model as 'SpurGear_25T_3.5m.CATPart'.

Result:

- A spur gear with 25 teeth, 3.5 mm module, and 20° pressure angle was successfully modeled in CATIA V5R21 using parametric formulas and standard 3D commands.

Experiment No. 2: Helical Spring Modeling Using CATIA V5R17

Aim:

- To **design and model a helical compression spring** using CATIA V5R17 with the help of helix and rib commands, and calculate spring parameters such as pitch, solid length, and spring diameters.

Software & Hardware Requirements:

- Software:
 - CATIA V5R17
 - Windows 10 or higher
- Hardware:
 - Minimum 8 GB RAM
 - Intel i5/i7 or AMD Ryzen Processor
 - Graphics Card supporting 3D CAD
 - Mouse with scroll and middle button

Theory:

- A helical compression spring stores mechanical energy and resists compression.
- It is defined by key parameters like:
 - D – Mean Coil Diameter
 - d – Wire Diameter
 - L_f – Free Length
 - L_s – Solid Length
 - n' – Number of Active Coils
 - p – Pitch (axial distance between adjacent coils)

Formulae Used:

1. Pitch (p) = $((L_f - L_s) / n') + d$
2. Solid Length (L_s) = $n' \times d$
3. Outer Diameter (D_o) = $D + d$
4. Inner Diameter (D_i) = $D - d$

Procedure in CATIA V5R17:

- **Step 1:** Open Part Design → Select YZ Plane → Enter Sketcher → Create a point 10 mm from the origin → Exit sketch → Named Sketch.1
- **Step 2:** In the command bar, type: C:HELIX → Enter Helix Parameters: Pitch = 4 mm, Height (L_f) = 40 mm, Orientation = Clockwise, Axis = Z-Axis, Starting Point = Sketch.1
- **Step 3:** Select ZX Plane → Enter Sketcher → Draw a circle with diameter = 2 mm at the bottom end of the helix → Exit Sketch → Named Sketch.2
- **Step 4:** Activate the Rib command: Select Sketch.2 as Profile, Select Helix curve as Center Curve, Choose Profile Control = Keep Angle → Click OK to create the helical spring

Observation & Calculations:

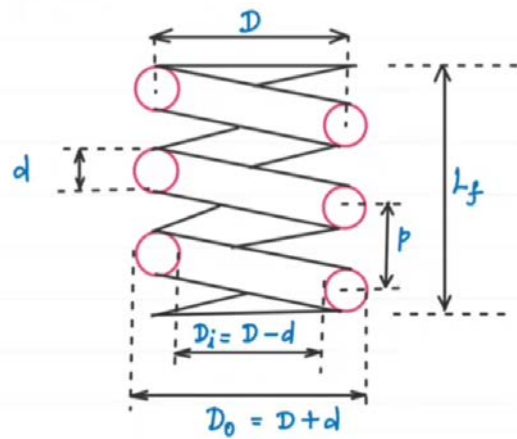
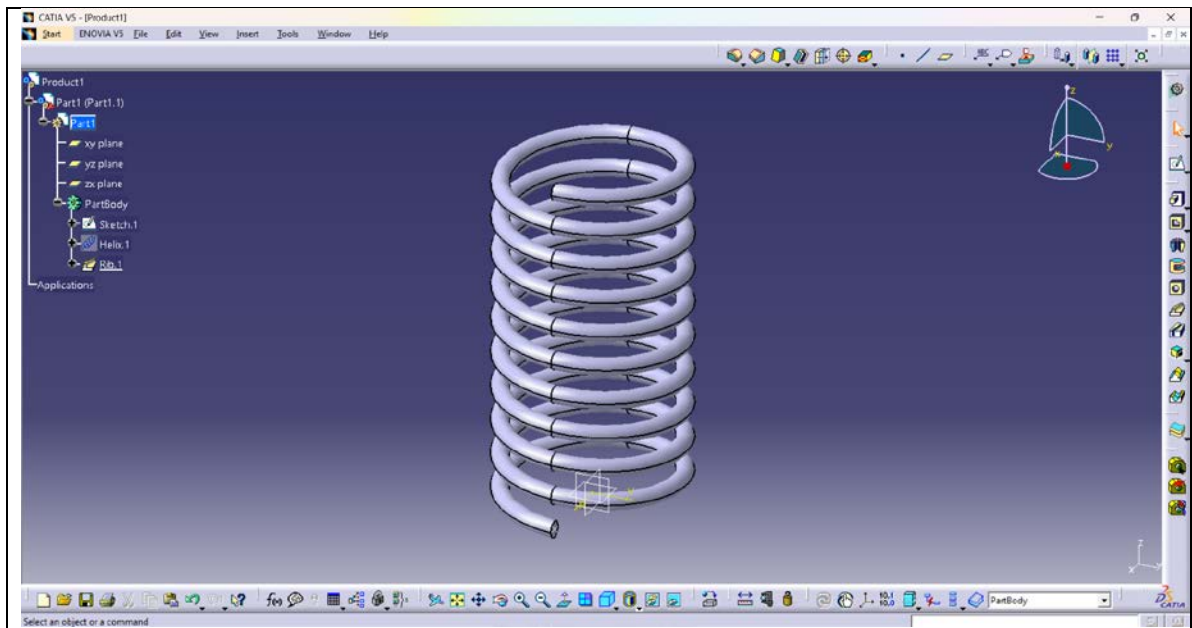
Parameter	Symbol	Value
Mean Coil Diameter	D	20 mm
Wire Diameter	d	2 mm
Free Length	L_f	40 mm
Pitch	p	4 mm
No. of Active Coils	n'	10
Solid Length	L_s	$10 \times 2 = 20$ mm
Outer Diameter	D_o	$20 + 2 = 22$ mm
Inner Diameter	D_i	$20 - 2 = 18$ mm

Verification:

$$p = \frac{L_f - L_s}{n'} + d = \frac{40 - 20}{10} + 2 = 4 \text{ mm.}$$

Result:

- A helical compression spring was successfully modeled in CATIA V5R17 using helix and rib features. The spring parameters such as pitch, solid length, and diameters were correctly calculated and verified.



pitch : $p = \frac{l_f - l_s}{n'}$

$D_o = D + d$

$D_i = D - d$

Experiment No. 3 – 3D Modeling of a Flywheel in CATIA V5R17

Aim:

- To design a 3D model of a flywheel using CATIA V5R17 software by applying Part Design and Sketch-based features including shaft, pad, pocket, multibody section, circular pattern, and fillet tools.

Hardware and Software Required:

Software:

- CATIA V5R21 or higher
- Windows 10 or above

Hardware:

- 8 GB RAM or more
- Intel i5/i7 or AMD Ryzen processor
- Graphics support for 3D CAD
- Mouse with scroll and middle button

Flywheel Parts:

1. Rim – The outer circular ring storing rotational energy
2. Hub – The central solid section that mounts onto the shaft
3. Spokes (Ribs) – The connecting arms between hub and rim (typically 6)
4. Keyway – A rectangular slot in the hub for shaft-key locking
5. Fillets – Smoothened edges at rib joints (for stress relief)

CATIA Commands Used:

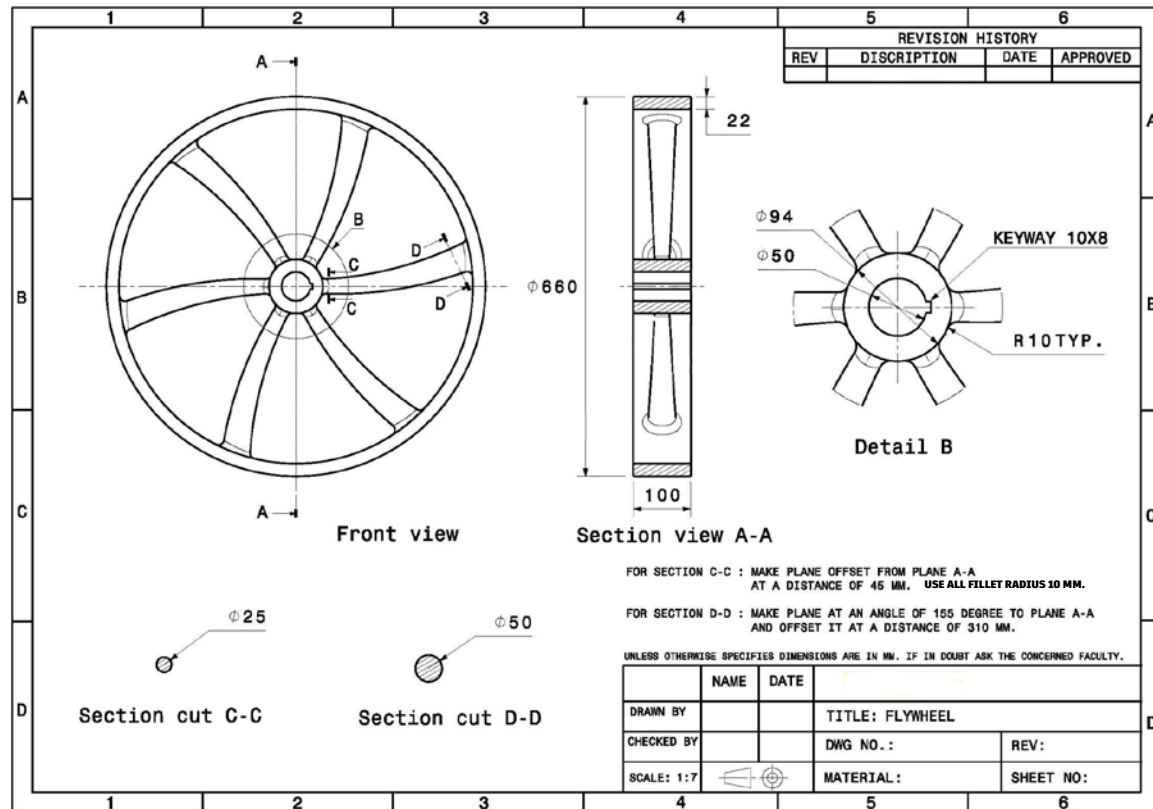
- Sketch
- Shaft
- Pad
- Pocket
- Reference Plane
- Multibody Section Solid
- Circular Pattern
- Fillet

Procedure:

1. **Open CATIA:** Start > Mechanical Design > Part Design. Enter part name: Flywheel.
2. **Create the Rim:**
 - Select YZ Plane > Sketch.
 - Draw vertical axis.
 - Draw a center rectangle measuring 22 mm × 100 mm, positioned 330 mm away from the origin.
 - Exit sketch and use Shaft tool to revolve 360° to form the rim.
3. **Create the Hub:**
 - Select XY Plane > Sketch.
 - Draw circle of 94mm diameter at origin.
 - Exit sketch and use Pad tool with 50mm thickness. Mirror extended.
4. **Create Keyway:**
 - On XY Plane, draw circle (50mm) and center rectangle (10x8mm).
 - Exit sketch > use Pocket tool, set length 50mm, mirror extended.
5. **Create Plane.1:**
 - Reference YZ Plane, offset 45mm.
 - On Plane.1, draw circle diameter 25mm.
 - Exit sketch.
6. **Create Plane.2 and Plane.3:**
 - Plane.2: Angle/Normal to Plane (YZ), angle -155°.
 - Plane.3: Offset 310mm from Plane.2 (reverse direction).
7. Draw circle (diameter 50mm) on Plane.3 and exit sketch.
8. Use **Multibody Section Solid:**
 - Select Sketches from Plane.1 and Plane.3.
 - Check arrows same direction, click OK.
9. Use **Circular Pattern:**
 - Object: **Spokes (Ribs).**
 - Complete crown, instances: 6, reference: Z-axis or hub face.
10. Apply 10mm **fillet** on 6 inner and outer rib ends.
11. **Save** the file: File > Save As > Flywheel_Model.CATPart.

Reference Drawing:

- The following image illustrates the flywheel design with dimensions and section views:



Result:

- A fully modeled flywheel was successfully created in CATIA V5R17 using 3D part modeling techniques including rim, hub, keyway, ribs, and fillets.

Experiment No. 4 – 3D Modeling of a Flanged Coupling in CATIA V5R17

Aim:

- To design and model an Unprotected Type Flanged Coupling using CATIA V5R17 by applying 3D part modeling tools such as Pad, Pocket, Shaft, Hole, Fillet, Pattern, and Assembly constraints.

Hardware and Software Requirements:

- Software:
 - CATIA V5R21 or higher
 - Windows 10 or above
- Hardware:
 - Minimum 8 GB RAM
 - Intel i5/i7 or AMD Ryzen processor
 - Dedicated graphics support for 3D CAD
 - Mouse with scroll wheel and middle button

Parts of Flanged Coupling:

1. Flanges (2 Nos.) – Circular plates to hold shaft ends.
2. Shafts (2 Nos.) – Cylindrical components transmitting torque.
3. Taper Keys (2 Nos.) – Locking elements between shaft and flange.
4. Hex Bolts (4 Nos.) – Fasteners to clamp the flanges together.
5. Hex Nuts (4 Nos.) – Locking nuts for the bolts.

CATIA Commands Used:

1. 2D Sketch: Point, Line, Circle, Rectangle, Circular Pattern
2. Workbench: Plane, Pad, Pocket, Shaft, Chamfer, Rib, Groove, Slot
3. Assembly Design: Constraints

Procedure:

Part 1 – Flange:

1. Open CATIA → Start > Mechanical Design > Part Design → Name it "Flange".
2. Select the ZY Plane → Sketch half of the front view of the flange, exit sketch, and revolve 360° using the Shaft tool.
3. Select the side face of the flange → Sketch the bolt pitch circle: Ø140 mm, and create 4 holes (Ø12 mm) at 90° spacing using the Circular Pattern tool.
4. Exit Sketch → Use Pad tool (Type: Up to Last) → OK.
5. Add a fillet (R3) to flange edges.
6. Select the side view of the flange → Sketch the key slot and shaft hole → Exit Sketch → Use Pad tool (Type: Up to Last) → OK.
7. Apply a draft to the key hole at angle (-0.573°) → Select the side face of the flange → Select the opposite face as the neutral plane → OK.
8. Save as "Flange.CATPart".

Part 2 – Shaft:

1. Create New Part → Name: "Shaft".
2. Sketch on the XY plane: Draw a cylinder Ø45 mm × 90 mm.
3. Use Pad to extrude.
4. Create a 14 mm wide keyway using Pocket tool.
5. Apply chamfer: 2 × 45° on one end.
6. Save as "Shaft.CATPart".

Part 3 – Taper Key:

1. Create New Part → Name: "TaperKey".
2. Sketch a rectangle: 80 mm × 14 mm.
3. Use Pad, then apply draft angle (1:100) to one side to create taper.

4. Apply draft to the key taper at 0.573° → Select the top face of the key → Select the opposite side as the neutral plane → OK.
5. Save as "Key.CATPart".

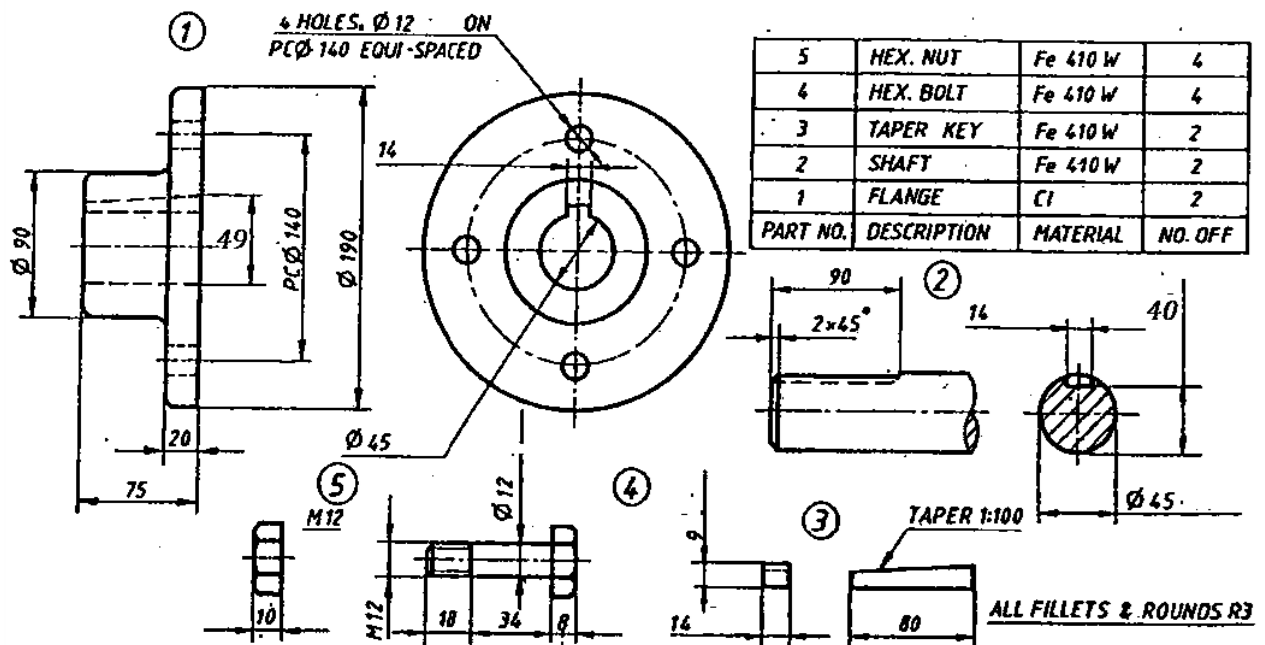
Part 4 – Hex Bolt and Nut:

1. Bolt: Model M12 diameter, 34 mm long threaded section using Helix tool, with 10 mm thick head.
2. Nut: Model hexagonal prism (10 mm thick) with M12 hole using Pocket tool; create internal threads with Helix tool.
3. Save as "Hex_Bolt.CATPart" and "Hex_Nut.CATPart".

Assembly of Flanged Coupling:

1. Open Assembly Design → New Product → Name: "Flange_Coupling".
2. Insert components: 2 × Flange, 2 × Shaft, 2 × Taper Key, 4 × Bolt, 4 × Nut.
3. Apply constraints:
 - Fix one flange.
 - Coincidence for shaft to flange hole.
 - Contact constraint for flange faces.
 - Coincidence for bolt holes.
 - Offset for spacing.
4. Save as "Flange_Coupling.CATProduct".

Reference Drawing: Refer to the provided engineering drawing of the flange coupling for dimensions and details.



All Dimensions in mm
Details of Flanged Coupling – Unprotected Type

Result:

- A fully modeled Unprotected Type Flanged Coupling was successfully created in CATIA V5R17 using 3D part modeling and assembly techniques. The assembly included flanges, shafts, keys, nuts, and bolts with proper constraints, alignment, and mechanical fit.

EXPERIMENT NO. 5: 3D MODELING OF A STRAP JOINT WITH GIB AND COTTER IN CATIA V5R17

Aim:

- To design and model a Strap Joint with Gib and Cotter using CATIA V5R17 by applying 3D part modeling tools such as Pad, Pocket, Fillet, chamfer, and Assembly constraints.

Hardware and Software Requirements:

❑ Software:

- CATIA V5R21 or higher
- Windows 10 or above

❑ Hardware:

- Minimum 8 GB RAM
- Intel i5/i7 or AMD Ryzen processor
- Dedicated graphics support for 3D CAD
- Mouse with scroll wheel and middle button

Parts of Strap Joint Assembly:

1. Rod – square shaft with slot to accommodate cotter.
2. Fork (Strap) – U-shaped strap to hold rod end.
3. Gib – Steel strip fitted alongside cotter to increase clamping and prevent loosening.
4. Cotter – Tapered locking element to secure rod to fork.

CATIA Commands Used:

1. 2D Sketch: Line, Rectangle, Trim
2. Workbench Tools: Pad, Pocket, Fillet, Chamfer
3. Assembly Design: Constraints, Fix, Coincidence, Contact

Procedure:

1. Part 1 – Fork (Strap)

- Create new part 'Fork'.
- Sketch U-profile per given dimensions (inner slot 25 mm width, arms 12.5 mm thick, total length 85.5 mm).
- Pad profile to given thickness.
- Add inner slot for rod and cotter clearance: 35.5 mm + 19 mm length.
- Apply R3 mm fillets to internal corners. Save as 'Fork.CATPart'.

2. Part 2 – Rod

- Open CATIA → Start → Mechanical Design → Part Design. Name it 'Rod'.
- On the XY plane, sketch a Rectangle: 25 mm; Pad to length 25 mm.
- Create a slot pocket: 7.5 mm wide, 35.5 mm long, located 19 mm from rod end.
- Apply fillets of R3 mm where required. Save as 'Rod.CATPart'.

3. Part 3 – Gib

- New part 'Gib'.
- Sketch rectangle 50 mm x 9.5 mm; Pad thickness 7.5 mm.
- Create offset notch: 8 mm x 9.5 mm cut.
- Apply R3 fillets to edges. Save as 'Gib.CATPart'.

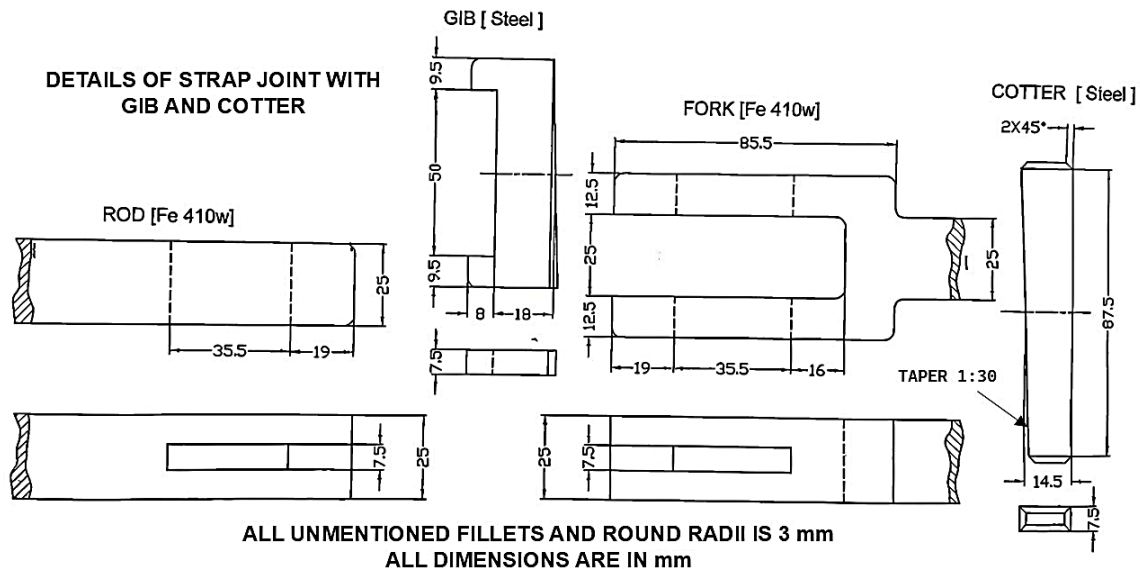
4. Part 4 – Cotter

- New part 'Cotter'.
- Sketch rectangle 87.5 mm x 25 mm; Pad thickness 14.5 mm.
- Apply taper 1:30 along length.
- Chamfer ends at $2 \times 45^\circ$. Save as 'Cotter.CATPart'.

5. Assembly of Strap Joint

- Open Assembly Design → New Product → Name: 'Strap_Assembly'.
- Insert parts: Rod, Fork, Gib, Cotter.
- Apply constraints:
 - Fix Fork.
 - Coincidence: Rod axis with Fork axis.
 - Contact: Rod shoulder with Fork inner face.
 - Insert Cotter through aligned slots.
 - Place Gib alongside Cotter.
- Ensure proper mechanical fit and alignment. Save as 'Strap_Assembly.CATProduct'.

Reference Drawing: Refer to the provided engineering drawing of a strap joint with gib and cotter for dimensions and details.



Result:

- A fully modeled Strap Joint with Gib and Cotter was successfully created in CATIA V5R17, including all components with proper dimensions, constraints, and assembly alignment.

Experiment No. 6 – 3D Modeling of a Knuckle Joint in CATIA V5R17

Aim:

- To design and model a Knuckle Joint using CATIA V5R17 by applying 3D part modeling tools such as Pad, Pocket, Fillet, Chamfer, Shaft, and Assembly constraints.

Hardware and Software Requirements:

- Software:
 - CATIA V5R21 or higher
 - Windows 10 or above
- Hardware:
 - Minimum 8 GB RAM
 - Intel i5/i7 or AMD Ryzen processor
 - Dedicated graphics support for 3D CAD
 - Mouse with scroll wheel and middle button

Parts of Knuckle Joint Assembly:

1. Fork End – Fork-shaped component with two arms.
2. Eye End – Single lug component with central hole.
3. Pin – Cylindrical pin to connect fork and eye.
4. Collar – Retaining element for pin.
5. Taper Pin – Locking pin to prevent axial movement of collar.

CATIA Commands Used:

1. 2D Sketch: Line, Circle, Rectangle, Trim
2. Workbench Tools: Pad, Pocket, Shaft, Fillet, Chamfer
3. Assembly Design: Constraints, Fix, Coincidence, Contact

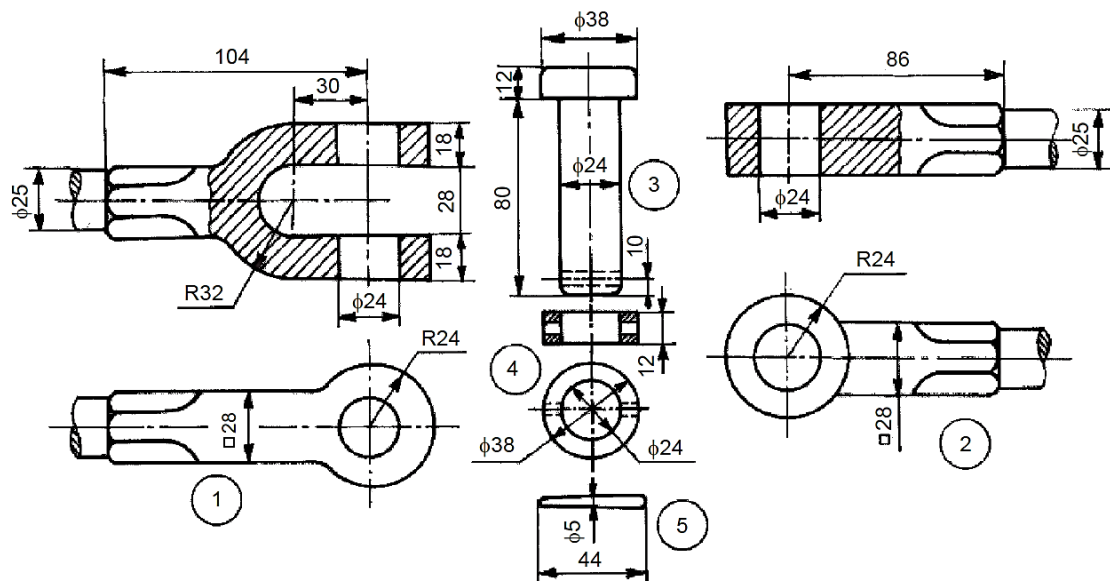
Procedure:

- **Part 1 – Fork End**
 - Create a new part named 'Fork_End'.
 - Sketch U-profile with central hole (Ø24 mm).
 - Pad to thickness 18 mm for each arm.
 - Apply fillets (R3 mm) to inner edges.
 - Save as Fork_End.CATPart.
- **Part 2 – Eye End**
 - New part: 'Eye_End'.
 - Sketch circular end with hole Ø24 mm.
 - Pad the shaft Ø25 mm to 86 mm length.
 - Apply R24 mm fillet to end profile.
 - Save as Eye_End.CATPart.
- **Part 3 – Pin**
 - New part: 'Pin'.
 - Sketch Ø24 mm x 80 mm cylinder.
 - Add Ø38 mm head, 12 mm thick.
 - Apply chamfer to edges.
 - Save as Pin.CATPart.
- **Part 4 – Collar**
 - New part: 'Collar'.
 - Sketch Ø38 mm x 12 mm ring with Ø24 mm hole.

- Pad to 12 mm thickness.
- Save as Collar.CATPart.
- **Part 5 – Taper Pin**
 - New part: 'Taper_Pin'.
 - Sketch $\varnothing 5$ mm \times 44 mm cylinder.
 - Apply taper to achieve locking.
 - Save as Taper_Pin.CATPart.
- **Assembly of Knuckle Joint**
 - Open Assembly Design \rightarrow New Product \rightarrow Name: 'Knuckle_Joint'.
 - Insert Fork_End, Eye_End, Pin, Collar, and Taper_Pin.
 - Apply constraints:
 - Fix Fork_End.
 - Coincidence: Eye_End hole axis with Fork_End hole axis.
 - Insert Pin through holes.
 - Place Collar at pin end.
 - Insert Taper_Pin through collar hole.
 - Ensure proper alignment and mechanical fit.
 - Save as Knuckle_Joint.CATProduct.

Reference Drawing:

- Refer to the provided engineering drawing of the knuckle joint for dimensions and details.



Parts list

Sl. No.	Name	Matl.	Qty.
1	Fork end	Forged steel	1
2	Eye end	Forged steel	1
3	Pin	Mild steel	1
4	Collar	Mild steel	1
5	Taper pin	Mild steel	1

Result:

- A fully modeled Knuckle Joint was successfully created in CATIA V5R17, including all components with proper dimensions, constraints, and assembly alignment.