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(Approved by AICTE, New Delhi. Affiliated to Anna University, Chennai)

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ANGUCHETTPALAYAM, PANRUTI – 607 106.

DEPARTMENT OF MECHANICAL ENGINEERING

CME 348 - MODERN ROBOTICS LAB

REGULATION - R-2021

THIRD YEAR - FIFTH SEMESTER

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INTROUCTION TO MODELING SOFTWARE PRO-E WILDFIRE 2.0

AIM:

To study about the important features, commands used in PRO/E WILDFIRE 2.0 modelling software.

INTRODUCTION:

What is Pro/ENGINEER?

Pro/ENGINEER, now known as **PTC Creo Parametric**, is a powerful **3D CAD (Computer-Aided Design)** software developed by PTC. It is widely used for product design, engineering, and manufacturing.

Key Features

1. **Parametric Modelling:** Designs are created with parameters and constraints, allowing easy modifications.
2. **3D Solid Modelling:** Create precise 3D models of parts and assemblies.
3. **Assembly Design:** Build complex assemblies from multiple components with relationships and constraints.
4. **Drafting and Detailing:** Generate 2D engineering drawings from 3D models.
5. **Simulation and Analysis:** Integrated tools for motion analysis, finite element analysis (FEA), and mechanism simulation.
6. **Manufacturing Integration:** Supports CNC machining and tooling design.
7. **Flexible Workflow:** Supports direct, parametric, and freestyle modelling.

Pro-Engineer is a high-end cad package. Pro-Engineer is a parametric solid modeler. It was developed by PARAMETRIC TECHNOLOGY CORPORATION. Pro-Engineer is a suite of programs that are used in the design, analysis, and manufacturing of a virtually unlimited range of products. Pro-Engineer develops models as solids, allowing us to work in a three-dimensional environment. These models have volume and surface area, so we can calculate mass properties directly from the geometry we create. Although we can manipulate their display on the screen, the model remains as solids. In a nutshell, Pro-Engineer is parametric, feature based solid modelling system.

FEATURE-BASED:

Pro-Engineer is a feature-based solid modelling tool. It builds the model using individual building blocks one at a time. We create our parts and assemblies by defining features like extrusions, sweeps, cuts, holes, slots, rounds, and so on, instead of specifying low-level geometry like lines, arcs and circles. Designers can think at a very high level and leave all the low-level geometric details for PRO/E WILDFIRE 2.0 to figure out. Features are specified by setting values of attributes such as reference planes or surfaces, direction of creation, pattern parameters, shape, dimensions and others.

Designing models in this manner is sometimes referred to as machinist theory because machinists create a single feature on the model at a time, with each manufacturing operation.

ASSOCIATIVE:

We can use Pro-Engineer to document models by creating parts, assemblies, and drawings. All these functions are fully associative-if we change a part in an assembly, the system automatically reflects that change in any other parts and drawings of that assembly. With this functionality we can change our design at any level, and the system reflects it all levels automatically.

PARAMETRIC:

“Parametric” means that the physical shape of the part or assembly is driven by the values assigned to the attributes of its features. We may define or modify a feature’s dimensions or other attributes at any time. Any changes will automatically propagate through our model. We can also relate the attributes of one feature to another. For example, if we are designing a new engine, the diameter of the cylinder will automatically change if we change the diameter of the piston.

SOLID MODELING:

“Solid Modelling” means that the computer model we create can contain all the information that a real solid object would have. It has volume and therefore, if we provide a value for the density of the materials, it has mass and inertia. The most useful thing about solid modelling is that it is impossible to create a computer model that is ambiguous or physically non-realizable.

PARENT CHILD RELATIONSHIPS:

An important aspect to feature-based modelling in PRO/E WILDFIRE 2.0 is the concept of parent-child relationships. A child feature is one that references a previously created parent feature. A parent feature will usually affect the child. For example, deleting a parent feature will delete all its children. PRO/E WILDFIRE 2.0 has special functions available to manage parent/child relationships.

DATUM PLANE CONSTRUCTION:**DATUM PLANE:**

A datum plane is representation of an infinite large planar surface at a user-defined orientation and location. Although the datum plane is infinite, it is displayed with boundaries that are continuously adjusted automatically to be slightly larger than the object. The boundary lines are colored Yellow or Red depending on which side of the plane we are viewing.

Plane: Insert > Datum > Plane

Default method creates three mutually perpendicular datum planes intersecting at the origin of the default co-ordinate system. These three planes represent XY, YZ and ZX. Planes are named as DTM1, DTM2 and DTM3.

DTM1 represents YZ-plane.

DTM2 represents ZX-plane.

DTM3 represents XY-plane.

DATUM AXIS:

Datum axis can be used as references for feature creation.

Datum axes are particularly useful for making datum planes, placing items concentrically and creating radial patterns.

PART MODELING:

PRO/E WILDFIRE 2.0 is a feature-based modelling software. Features in PRO/E WILDFIRE 2.0 are of two kinds- pick and place features, sketched features. Holes, rounds and chamfers are examples of pick and place features. Extrude, revolve, sweep and blend are examples of sketched features.

SKETCHED FEATURES:

PROTRUSION FEATURE:

Protrusion is the method of adding a solid material to the model. PRO/E WILDFIRE 2.0 provides the following basic methods of adding material to a model.

EXTRUDE	Creates a solid feature by extruding a section normal to the section plane
REVOLVE	Creates a solid feature by revolving a section about an axis.
SWEEP	Creates a solid feature by sweeping a section about a trajectory.
BLEND	Creates a solid feature by blending various cross sections at various levels.

EXTRUSION:

Feature > Create > Solid > Protrusion > Extrusion.

ATTRIBUTE TYPES:

ONE SIDE: Adds the material in one side of the cross section only.

BOTH SIDES: Adds the material on both sides of the cross section.

DEFINING SKETCHING PLANE:

We must specify the sketch plane to draw the cross section. To define the sketch plane, we can use the existing datum plane, existing planer face of the solid or we can create a new plane sketch.

DIRECTION OF FEATURE CREATION:

After specifying the sketch plane, we must specify the direction of extrusion. By default, PRO/E WILDFIRE 2.0 assumes the positive side of the sketch plane as feature direction. Here PRO/E WILDFIRE 2.1 will prompt for extrusion direction. We can either accept the default direction or flip the direction.

ORIENTATING THE SKETCH PLANE:

Once the direction of extrusion is defined PRO/E WILDFIRE 2.0 will prompt for orienting the sketch plane. Orientation plane can lie horizontal or vertical in the section view. Since the orientation plane as yellow and red side, we must specify the direction of the yellow side as Top, Bottom, Left or Right. Once the orientation plane is defined, PRO/E WILDFIRE 2.0 automatically changes the view as normal view sketch plane. Once the orientation plane is defined, tools needed for sketching the cross section will be provided by PRO/E WILDFIRE 2.0.

SKETCH REFERENCE:

PRO/E WILDFIRE 2.0 automatically selects two mutually perpendicular planes which are normal to the sketching plane as horizontal and vertical references for dimensioning and constraining the sketch.

EXTRUSION DEPTH:

BLIND	Enter a dimension for the feature depth. We can then control the feature depth by changing the depth dimension.
2 SIDE BLIND	For a feature defined as both sides, enter a separate depth value for each.
THRU NEXT	Terminate the feature at the next part surface.
THRU ALL	The new feature intersects all surfaces.
THRU UNTIL	Extend the feature until the intersection.
UP TO CURVE	Specify the depth up to the plane parallel to the sketching plane, and passing through the selected edge, axis or datum curve.
UP TO SURFACE	Specify the depth up to a selected surface.

REVOLVE:

Feature > Create > Solid > Protrusion > Revolve

The revolve option creates a feature by revolving the sketched section around the sketched centreline. When sketching the section, the first centreline that is sketched is used as the axis of revolution. The sketched section must lie completely on one side of the centreline, and the section must be closed. The section can be revolved to one or both side of the sketching plane and the angle of revolution are specified by the user. The angle values of revolution can be specified as follows:

VARIABLE	Specify angle less than 360 degree.
90 DEGREE	90 Degree revolution
180 DEGREE	180 Degree revolution
270 DEGREE	270 Degree revolution
360 DEGREE	360 Degree revolution
UPTO PNT/VERTEX	Revolve up to a datum plane point or model vertex.
UPTO PLANE	Revolve up to an existing plane or model planar surface that contains the axis of revolution.

RIB:

Feature > Create > Solid > Rib

A rib is a special type of protrusion designed to create a thin wall or web to support two surfaces. The rib is used to increase the strength of the part. There are two types of ribs-straight and rotational ribs.

HOLE:

Feature > Create > Solid > Hole

All straight holes are created with constant diameter. A hole feature always removes material from our model. Placement references selected to locate the hole on our model depend on the placement type location options.

ROUND:

Features > Create > Solid > Round

Round features create fillet or rounded smooth transition with either circular or conic profile between adjacent surfaces. An edge round smoothest the hard edges between adjacent surfaces. Round features can add or remove material depending on the edge references selected for an edge round are the model edge or edges.

SWEEP:

Feature > Create > Solid > Protrusion > Sweep

A sweep is created by defining two sections. The first section is the trajectory and the second is the cross section. The trajectory is always created as the first step. The cross section is created and located with respect to the trajectory. Sweep can be used as forms for protrusions, cuts and slots.

BLEND:

Feature > Create > Solid > Protrusion > Blend

Blending is the methods of defining a volume by connecting series of at least two planar sections together at their edges with transitional surfaces.

PATTERN:

Feature > Pattern

Patterns are multiple features created from a single feature that behaves as one feature.

ASSEMBLY CREATION:

Assembly mode in PRO/E WILDFIRE 2.0 enables us to place component parts and sub-assemblies together to form assemblies, as well as to design parts based on how they should fit together. We can then modify, analyse, or re-orient the resulting assemblies.

The placement commands used to assemble the component part or sub-assembly into the assembly are defined as follows:

MATE: Selected surface point in opposite direction and become co-planar

MATE OFFSET: Selected surface point in opposite directions and are offset by a specified value. The offset value can be modified to provide design flexibility.

ALIGN: Selected surface points in the same direction and are co-planar axis may be made co-axial with the align command.

ALIGN OFFSET: Selected surface point in the same direction and are offset by a specified value which can be modified to provide design flexibility.

ORIENT: The selected planar surfaces are made parallel and point in the same direction.

INSERT: Makes two surfaces of revolution coaxial.

COMMON PRO/E COMMANDS AND THEIR USES

1. File Commands

- **New** — Create a new part, assembly, or drawing.
- **Open** — Open an existing file.
- **Save** — Save the current file.
- **Save As** — Save the file with a new name.
- **Import/Export** — Bring in or send out CAD data (e.g., STEP, IGES).

2. Sketching Commands

- **Sketch** — Enter sketch mode to create 2D profiles.
- **Line** — Draw straight lines.
- **Circle** — Draw circles.
- **Rectangle** — Draw rectangles.
- **Arc** — Draw arcs.
- **Dimension** — Apply dimensions and constraints.
- **Trim** — Cut unwanted parts of sketch geometry.
- **Pattern** — Create repeated sketch elements.

3. Feature Creation Commands

- **Extrude** — Create a 3D feature by extending a sketch.
- **Revolve** — Create a 3D feature by rotating a sketch around an axis.
- **Hole** — Create holes with defined parameters.
- **Chamfer** — Create beveled edges.
- **Fillet** — Round off edges.
- **Shell** — Hollow out a solid body.
- **Pattern** — Create repeated features.
- **Mirror** — Reflect features or geometry about a plane.

4. Assembly Commands

- **Assemble** — Insert components into assembly.
- **Mate** — Align surfaces or faces.
- **Align** — Align edges or axes.
- **Insert Constraint** — Position parts relative to each other.
- **Explode** — Create exploded views.
- **Check Interference** — Detect collisions between parts.

5. View and Visualization Commands

- **Zoom** — Zoom in/out.
- **Pan** — Move view.
- **Rotate** — Rotate the model.
- **Display Style** — Switch between wireframe, shaded, hidden line, etc.
- **Section View** — Create cross-sectional views.
- **Measure** — Measure distances, angles, radii.

6. Simulation Commands (Mechanism Mode)

- **Mechanism** — Enter mechanism design mode.
- **Insert Joint** — Define prismatic, revolute, or spherical joints.
- **Add Motor** — Assign motors to joints.
- **Run Analysis** — Perform kinematic or dynamic simulation.
- **Playback** — Visualize motion.
- **Export Animation** — Save simulation video.

7. Drafting Commands

- **Create Drawing** — Start a new 2D drawing from 3D model.
- **Place Views** — Insert front, top, side, isometric views.
- **Dimensioning** — Add dimensions to drawing.
- **Annotations** — Add notes, symbols, GD&T.
- **Bill of Materials** — Generate BOM tables.

GENERAL PROCEDURE FOR USING PRO/ENGINEER (PRO E) SOFTWARE

Step 1: Start Pro/E and Set Up Your Workspace

- Launch Pro/ENGINEER (Creo Parametric).
- Set working directory/project folder.
- Choose units and system options if needed.

Step 2: Create a New Part

- Go to **File > New > Part**.
- Name the part and select modelling style (Solid, Surface, etc.).
- Click **OK**.

Step 3: Create a 2D Sketch

- Select a plane (Front, Top, Right).
- Enter **Sketch Mode**.
- Use sketch tools (Line, Circle, Rectangle) to draw profile.
- Apply dimensions and constraints to fully define the sketch.

Step 4: Create 3D Features from Sketch

- Exit sketch mode.
- Use features like **Extrude**, **Revolve**, or **Sweep** to create 3D shape.
- Add additional features (holes, fillets, chamfers) as needed.

Step 5: Save the Part

- Use **File > Save** to save the part file.

Step 6: Create Additional Parts (if needed)

- Repeat Steps 2-5 for other components.

Step 7: Create an Assembly

- Go to **File > New > Assembly**.
- Name the assembly.
- Insert the base part and fix it in space.

- Insert other parts and constrain them using **Mate**, **Align**, and **Insert** constraints.
- Use appropriate constraints to define relative positions and allow intended motion (if any).

Step 8: Perform Motion Simulation (Optional)

- Switch to **Mechanism Mode** from Applications menu.
- Define joints (revolute, prismatic, spherical) between parts.
- Add motors or actuators to joints.
- Set simulation parameters (time, velocity).
- Run simulation and analyze results.

Step 9: Create Drawings (Optional)

- Go to **File > New > Drawing**.
- Select standard or custom templates.
- Place views from parts or assembly (front, top, side, isometric).
- Add dimensions, annotations, and notes.
- Save the drawing file.

Step 10: Export / Share Files

- Export models or drawings in standard formats like STEP, IGES, or PDF.
- Share files for manufacturing or collaboration.

TYPICAL APPLICATIONS

- Mechanical part and assembly design
- Industrial product design
- Automotive and aerospace components
- Robotics and automation equipment
- Consumer products

ADVANTAGES

- Robust parametric design for easy updates
- Comprehensive toolset from concept to manufacturing
- Strong simulation and validation capabilities
- Integration with PLM (Product Lifecycle Management) systems
- Large user base and industry acceptance

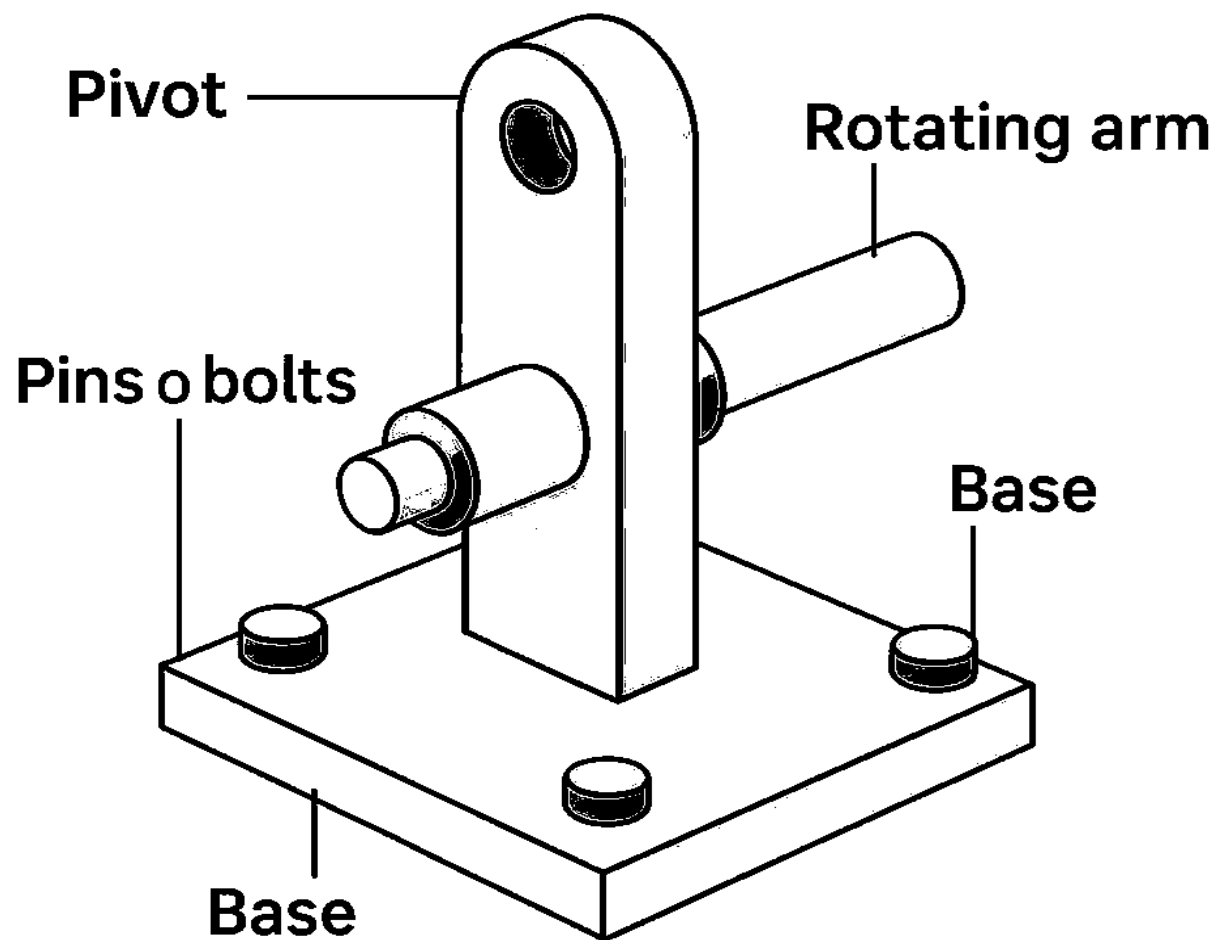
BASIC WORKFLOW IN PRO/E

1. **Sketching:** Create 2D profiles.
2. **Feature Modelling:** Extrude, revolve, cut, and modify to build 3D shapes.
3. **Assembly:** Combine parts and define constraints.
4. **Simulation:** Test motion, stress, and other factors.
5. **Drawing:** Generate manufacturing drawings.
6. **Export:** Output models for CAM or 3D printing.

RESULT:

Thus, the important features and commands used in PRO/E WILDFIRE 2.0 Modelling software were studied.

Rotationl joint assembly



Ex. No: 01

Date:

3D MODELLING AND MOTION SIMULATION OF ROTATIONAL JOINT

ASSEMBLY

AIM:

To Design 3D modelling and motion simulation of rotational joint assembly using CAD software.

PROCEDURE:

Part 1: 3D Modelling of Components in Pro/E

Before creating the assembly, each component (e.g., base, shaft, pin, rotating arm) must be individually modelled.

1. Open Pro/E (Creo)

- Start the software.
- Select **New > Part > Solid > Create**.

2. Model Each Part Individually

Repeat this for each component:

- Sketch the base shape (extrude, revolve, etc.).
- Use tools like Extrude, Revolve, Hole, Round, Chamfer, etc.
- Save each part file.

Example:

- Part1: Base (fixed)
- Part2: Rotating arm
- Part3: Shaft or pin

Part 2: Assembly of Rotational Joint

1. Create New Assembly

- File > New > Assembly > Solid > Create.

2. Place Base Component

- Use **Component > Assemble**.
- Select the base part.
- Fix it using constraints (Default or Fully Constrain).

3. Assemble Rotating Arm

- Use **Mate, Align, Insert** constraints:
- Align the shaft of the arm to the hole in the base.
- Allow rotation about the axis by avoiding over-constraining.

- The rotating part should be free to rotate around the defined axis (one degree of freedom).

Part 3: Motion Simulation (Mechanism Module)

1. Switch to Mechanism Application

- Go to **Applications > Mechanism**.

2. Define Joint

- Select **Insert > Joint**.
- Choose **Revolute Joint** for rotation:
- Select axis of rotation on both parts.
- Ensure that only rotational motion is allowed.

3. Add Motor (Optional for Simulation)

- Mechanism > Motors
- Type: **Servo Motor**
- Select rotational axis
- Define:
 - Type: Constant/Servo Profile
 - Value: RPM or angular velocity

4. Run Simulation

- Use **Mechanism > Analysis > New**:
- Type: **Kinematic** or **Dynamic** (if forces are defined).
- Set time and frame rate.
- Click **Run** to simulate motion.

5. Observe Results

- Use **Playback** or **Animations** to visualize the rotation.
- Export animation if needed.

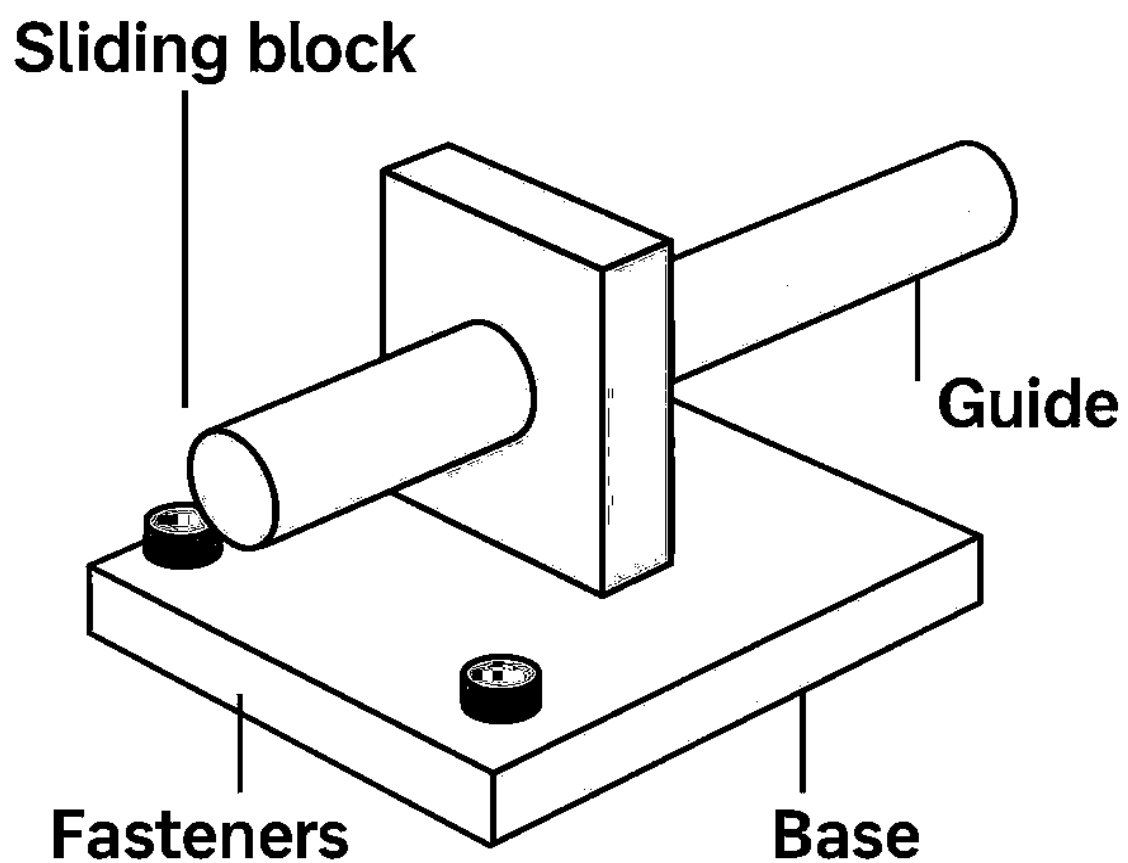
Saving & Export

- Save your assembly and simulation results.
- You can export animation as a video (via Mechanism > Playback > Save As AVI).

RESULT:

Thus, a rotational joint assembly was designed, a motion study was performed, and an animation of the assembly was created.

PRISMATIC JOINT ASSEMBLY



Ex. No: 02

Date:

3D MODELLING AND MOTION SIMULATION OF PRISMATIC JOINT

ASSEMBLY

AIM:

To Design 3D modelling and motion simulation of prismatic joint assembly using CAD software.

PROCEDURE:

Part 1: 3D Modelling of Components

Step 1: Create Base Part (Guide)

- Open Pro/E.
- **New > Part > Solid > Create.**
- Sketch the base profile (e.g., a rectangular block with a groove/slot for slider).
- Use **Extrude** or **Cut** to form the guideway.
- Add fillets or chamfers as needed.
- Save the file as **Base.prt**.

Step 2: Create Slider Part

- New > Part > Solid > Create.
- Sketch the slider profile (should fit inside the guide groove).
- Use **Extrude** to create slider block.
- Add features such as holes or bosses if needed.
- Save as **Slider.prt**.

Part 2: Assembly of Prismatic Joint

Step 1: Create New Assembly

- **File > New > Assembly > Solid > Create.**
- Insert **Base.prt** first.
- Fix the base in space (default).

Step 2: Insert Slider

- **Component > Assemble > Select Slider.prt.**
- Position roughly inside the guide slot.
- Use constraints to:
- Align the slider to the groove faces (Mate or Align constraints).
- Do NOT fully constrain translation along the groove axis — allow movement in that direction.

- Make sure rotation and other translations are restricted (only 1 DOF: linear sliding).

Part 3: Motion Simulation in Mechanism Module

Step 1: Switch to Mechanism Mode

- Applications > Mechanism.

Step 2: Define Prismatic Joint

- Insert > Joint > Prismatic Joint.
- Select the axis/edge on the base that defines the guide direction.
- Select the corresponding face or edge on the slider.
- Confirm joint allows sliding motion only (one translational DOF).

Step 3: Add Linear Motor (Optional)

- Mechanism > Motors > Add Linear Motor.
- Select the slider component.
- Define:
 - Direction of motion (along sliding axis).
 - Velocity or displacement profile (constant velocity or custom).

Step 4: Run Simulation

- Mechanism > Analysis > New.
- Select Kinematic Analysis.
- Define simulation time.
- Run simulation to observe the slider moving linearly.

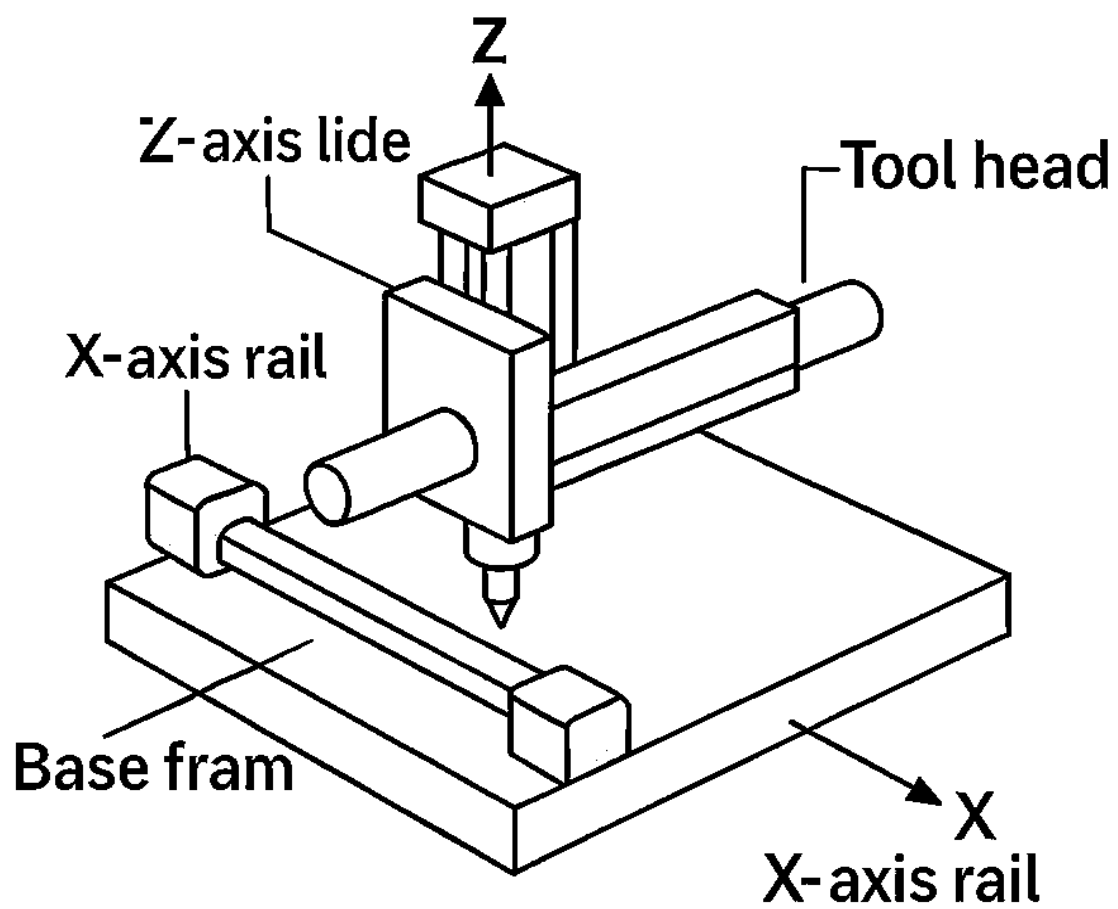
Step 5: View and Export

- Use playback tools to visualize the slider movement.
- Export animation if desired (Mechanism > Playback > Save As AVI).

RESULT:

Thus, the prismatic joint assembly was designed, a motion study was performed, and an animation of the assembly was created.

CARTESIAN ROBOT



Ex. No: 03

Date:

3D MODELLING AND MOTION SIMULATION OF CARTESIAN ROBOT

AIM:

To Design 3D modelling and motion simulation of cartesian robot using CAD software.

PROCEDURE:

Part 1: 3D Modelling of Cartesian Robot Components

A Cartesian robot generally consists of:

- Base frame (fixed)
- Linear guides/slides for X, Y, and Z axes
- Carriages/sliders moving along those guides
- End effector/tool at the end

Step 1: Model Base Frame

- New > Part > Solid.
- Sketch and extrude a base frame (block or plate).
- Save as **Base.prt**.

Step 2: Model X-axis Guide and Slider

- Create a guide rail/profile (long rectangular extrusion).
- Create a slider that will move along X-axis guide.
- Save as **X_Guide.prt** and **X_Slider.prt**.

Step 3: Model Y-axis Guide and Slider

- Model Y guide fixed to the X slider.
- Model Y slider that moves along the Y guide.
- Save as **Y_Guide.prt** and **Y_Slider.prt**.

Step 4: Model Z-axis Guide and Slider (Vertical Axis)

- Model vertical guide fixed to Y slider.
- Model Z slider that moves up/down.
- Save as **Z_Guide.prt** and **Z_Slider.prt**.

Step 5: Model End Effector

- Create the tool or gripper.
- Attach it to the Z slider.
- Save as **End_Effector.prt**.

Part 2: Assembly of Cartesian Robot

Step 1: Create New Assembly

- New > Assembly > Solid.

Step 2: Insert Base Frame

- Fix it in space.

Step 3: Assemble X Guide and Slider

- Insert X_Guide onto Base with constraints.
- Insert X_Slider onto X_Guide:
- Constrain slider to translate along X-axis only.

Step 4: Assemble Y Guide and Slider

- Insert Y_Guide onto X_Slider.
- Insert Y_Slider onto Y_Guide:
- Constrain to translate along Y-axis only.

Step 5: Assemble Z Guide and Slider

- Insert Z_Guide onto Y_Slider.
- Insert Z_Slider onto Z_Guide:
- Constrain to translate along Z-axis only.

Step 6: Attach End Effector

- Assemble End Effector onto Z_Slider.
- Fully constrain it relative to Z slider.

Part 3: Motion Simulation in Mechanism Module

Step 1: Switch to Mechanism Mode

- Applications > Mechanism.

Step 2: Define Prismatic Joints

- For each sliding pair (X, Y, Z), insert a Prismatic Joint allowing translation only along the respective axis.
- Select corresponding edges or faces on guide and slider for each joint.

Step 3: Add Linear Motors

- For each axis, add a **Linear Motor**:
- Define velocity/displacement profiles.
- Control movement along X, Y, and Z.

Step 4: Run Kinematic Analysis

- Create a new **Kinematic Analysis**.
- Set simulation time.
- Run to observe coordinated motion of the Cartesian robot.

Step 5: Visualize and Export

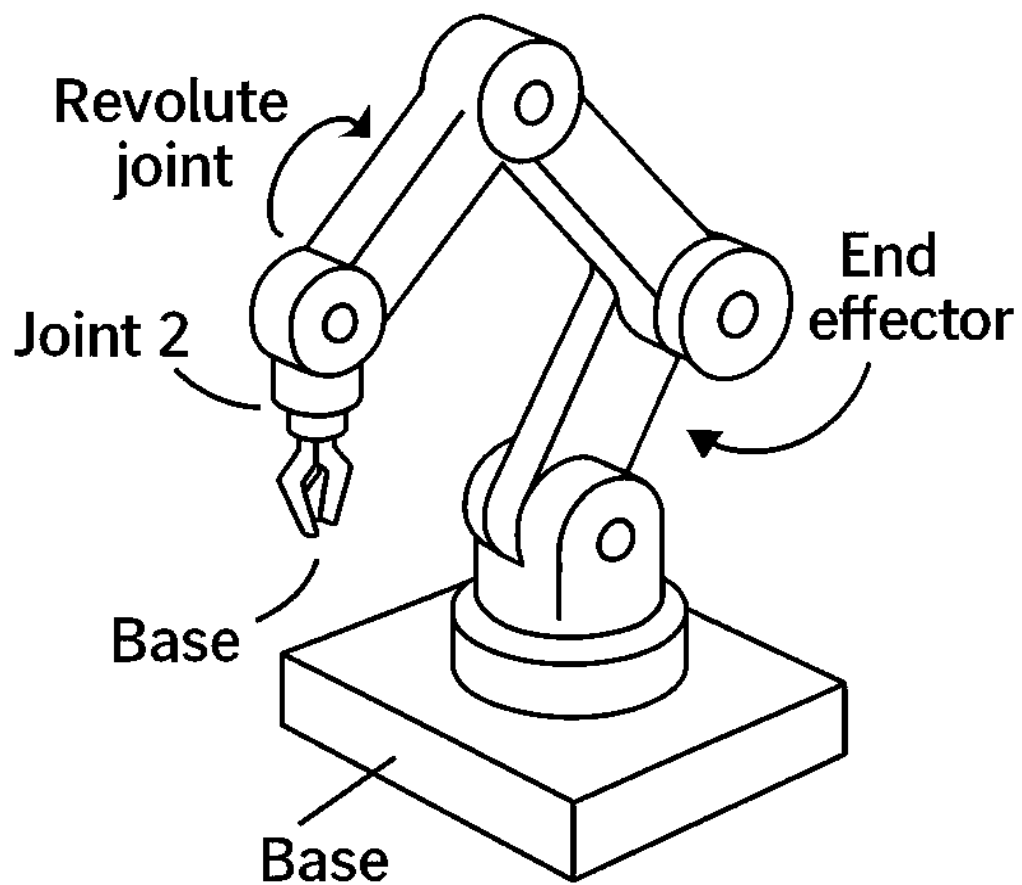
- Use playback tools to see the robot move along all three axes.

- Export animations if needed.

RESULT:

Thus, the cartesian robot assembly was designed, a motion study was performed, and an animation of the assembly was created.

ARTICULATED ROBOT



Ex. No: 04

Date:

3D MODELLING AND MOTION SIMULATION OF ARTICULATED ROBOT

AIM:

To Design 3D modelling and motion simulation of articulated robot using CAD software.

PROCEDURE:

Part 1: 3D Modelling of Robot Components

Articulated robots typically have multiple links connected by rotational (revolute) joints.

Step 1: Model Base

- New > Part > Solid.
- Sketch and extrude a sturdy base.
- Save as Base.prt.

Step 2: Model Robot Links

For each link:

- New > Part > Solid.
- Sketch profile (beam, arm segment).
- Extrude to create the link.
- Add features like holes or bosses for joint connection.
- Save as Link1.prt, Link2.prt, etc.

Step 3: Model Joints (Pins, Shafts)

- Model joint parts if needed (pins, bearings).
- Save as separate parts or model as features on links.

Part 2: Assembly of Articulated Robot

Step 1: Create New Assembly

- File > New > Assembly > Solid > Create.

Step 2: Insert Base

- Fix base in space.

Step 3: Assemble First Link to Base

- Insert Link1.
- Use **Insert > Mate/Align/Insert** constraints to connect Link1 to Base at joint location.
- Leave rotation about joint axis free (do not fully constrain).

Step 4: Assemble Subsequent Links

- Insert Link2, Link3, etc.
- Connect each link to the previous link using mate and align constraints at joint points.

- Ensure rotational degree of freedom about the joint axis remains free.

Part 3: Define Rotational Joints and Motion Simulation

Step 1: Switch to Mechanism Mode

- Applications > Mechanism.

Step 2: Define Revolute Joints

- For each joint, insert Revolute Joint:
- Select the rotation axis on adjacent links.
- Confirm only rotation about this axis is allowed.

Step 3: Add Motors to Joints

- For each revolute joint, add a Servo Motor or Constant Velocity Motor:
- Specify angular velocity or angular displacement.
- Set a motion profile if needed.

Step 4: Run Kinematic or Dynamic Analysis

- Mechanism > Analysis > New.
- Choose Kinematic (no forces) or Dynamic (forces and masses considered).
- Set simulation time.
- Run simulation.

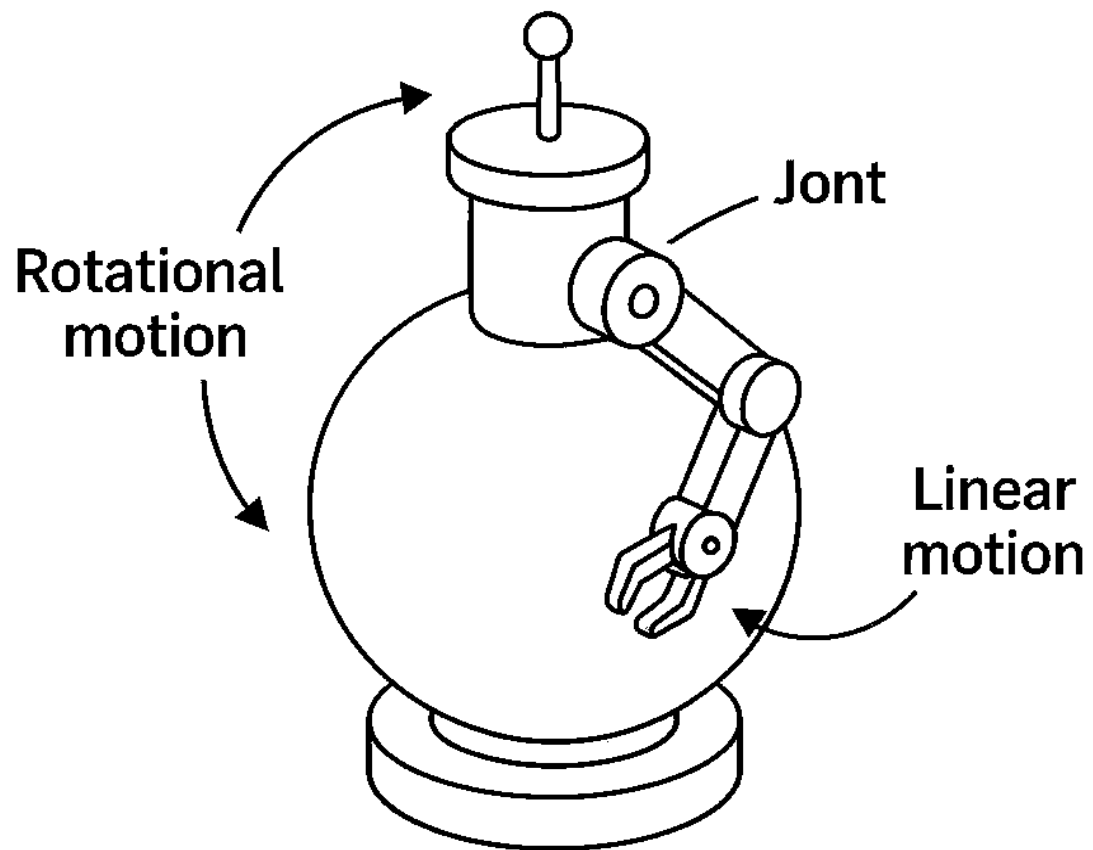
Step 5: Observe Motion

- Use playback tools to visualize articulated arm movement.
- Verify joint angles and link motions.
- Export animation or plot joint variables if needed.

RESULT:

Thus, the articulated robot assembly was designed, a motion study was performed, and an animation of the assembly was created.

SPHERICAL ROBOT



Ex. No: 05

Date:

3D MODELLING AND MOTION SIMULATION OF SPHERICAL ROBOT

AIM:

To Design 3D modelling and motion simulation of spherical robot using CAD software.

PROCEDURE:

Part 1: 3D Modelling of Components

Step 1: Model Base or Fixed Support

- New > Part > Solid.
- Sketch and extrude the base or fixed support structure.
- Save as Base.prt.

Step 2: Model Robot Links

- Create each arm segment or link as a separate part.
- Model spherical joints if needed (e.g., ball shapes, sockets).
- Save as Link1.prt, Link2.prt, etc.

Part 2: Assembly of Spherical Robot

Step 1: Create New Assembly

- File > New > Assembly > Solid > Create.

Step 2: Insert Base and Fix

- Insert Base and fix it in space.

Step 3: Insert First Link and Define Joint

- Insert Link1.
- Use constraints (Mate, Align) to position the link relative to the base.
- Leave rotational freedom to simulate ball-and-socket joint (allow rotation in multiple axes).

Step 4: Insert Subsequent Links

- Insert Link2, etc.
- Assemble to previous link with appropriate constraints.
- Model spherical joints by combining multiple revolute joints or universal joints allowing 3 DOF rotations.

Part 3: Motion Simulation (Multi-DOF Joint Simulation)

Step 1: Switch to Mechanism Mode

- Applications > Mechanism.

Step 2: Model Spherical Joint

- Pro/E does not have a direct spherical joint, but you can create it by combining:
- Three Revolute Joints with perpendicular axes intersecting at the joint center.
- Or a Universal Joint plus additional revolute joint.

Step 3: Add Motors

- Add **Servo Motors** or **Constant Velocity Motors** to each revolute joint axis.
- Define motion profiles for desired rotation sequences.

Step 4: Run Kinematic or Dynamic Analysis

- Mechanism > Analysis > New.
- Select type (Kinematic or Dynamic).
- Set simulation parameters.
- Run simulation to see the spherical motion.

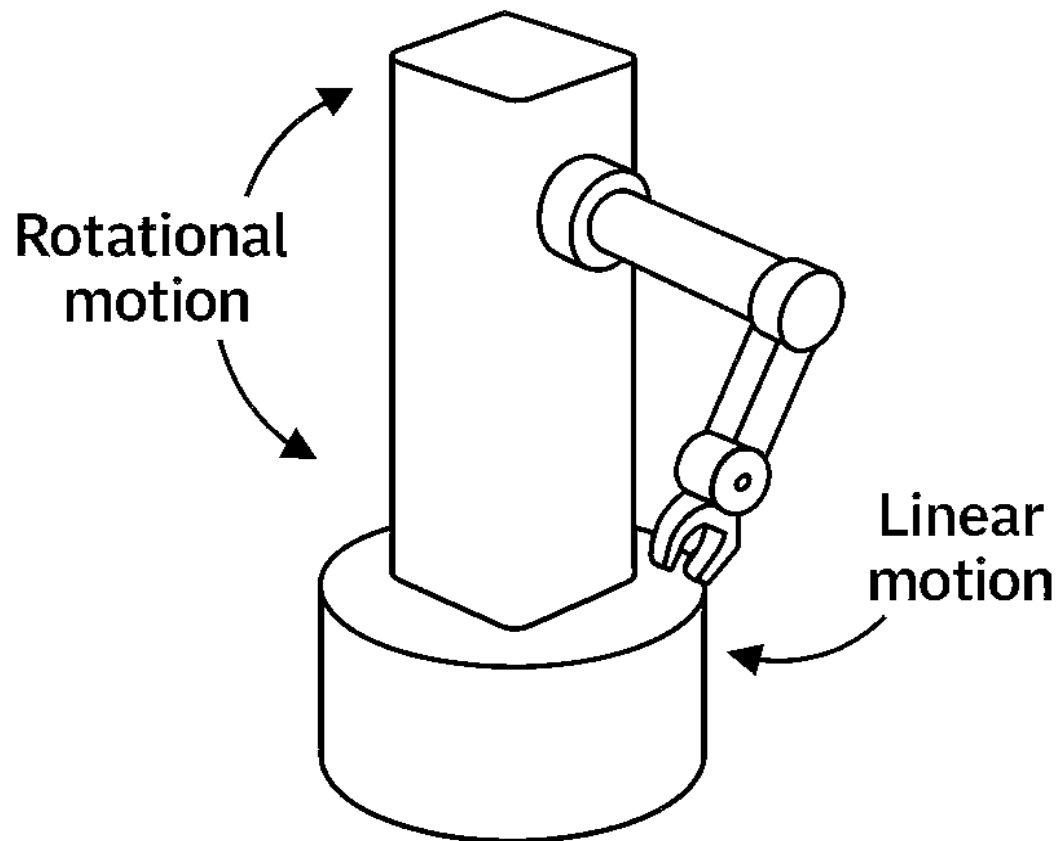
Step 5: Visualize

- Playback to see the compound rotations.
- Export animation if needed.

RESULT:

Thus, the spherical robot assembly was designed, a motion study was performed, and an animation of the assembly was created.

CYLINDRICAL ROBOT



Ex. No: 06

Date:

3D MODELLING AND MOTION SIMULATION OF CYLINDRICAL ROBOT

AIM:

To Design 3D modelling and motion simulation of cylindrical robot using CAD software.

PROCEDURE:

Part 1: 3D Modelling of Components

Step 1: Model Base (Fixed)

- New > Part > Solid > Create.
- Sketch and extrude the robot base (a platform or column).
- Save as **Base.prt**.

Step 2: Model Rotational Arm (Rotates around vertical axis)

- New part.
- Sketch and extrude a cylindrical arm or disk that will rotate.
- Save as **Rotating_Arm.prt**.

Step 3: Model Vertical Slide (Prismatic Joint)

- New part.
- Sketch and extrude a slider or column that moves vertically along the rotating arm.
- Save as **Vertical_Slider.prt**.

Step 4: Model End Effector (Tool or Gripper)

- New part.
- Model the end effector attached to the vertical slider.
- Save as **End_Effector.prt**.

Part 2: Assembly of Cylindrical Robot

Step 1: Create New Assembly

- File > New > Assembly > Solid > Create.

Step 2: Insert Base and Fix

- Insert **Base.prt**.
- Fix it in space.

Step 3: Assemble Rotational Arm on Base

- Insert **Rotating_Arm.prt**.
- Constrain it so it can rotate about the vertical axis (use Insert or Mate/Align constraints allowing rotation about Z-axis, but restricting other DOFs).

Step 4: Assemble Vertical Slider on Rotating Arm

- Insert **Vertical_Slider.prt**.

- Constrain it so it can slide (prismatic joint) along the vertical axis of the rotating arm.
- Allow translation only along Z-axis.

Step 5: Attach End Effector

- Assemble End_Effector.prt to the vertical slider.
- Fully constrain relative to vertical slider.

Part 3: Motion Simulation in Mechanism Module

Step 1: Switch to Mechanism Mode

- Applications > Mechanism.

Step 2: Define Joints

- Insert Revolute Joint between Base and Rotating Arm:
- Select vertical axis to allow rotation.
- Insert Prismatic Joint between Rotating Arm and Vertical Slider:
- Select vertical axis for sliding.

Step 3: Add Motors

- Add Rotational Motor to the revolute joint.
- Define angular velocity or position profile.
- Add Linear Motor to the prismatic joint.
- Define velocity or displacement profile.

Step 4: Run Simulation

- Mechanism > Analysis > New > Kinematic.
- Set simulation time.
- Run to observe combined rotational and linear motion.

Step 5: Visualize and Export

- Use playback controls to watch the robot move.
- Export animation if needed.

RESULT:

Thus, the cylindrical robot assembly was designed, a motion study was performed, and an animation of the assembly was created.

