Introduction to Mechanism Design

The optional **Mechanism Design** module of Pro/ENGINEER **Wildfire** is a kinematics design tool. This tool can be used to:

- Create simple and complex mechanical mechanisms.
- Define connections between assembly components.
- Define servo motors and various parameters for them.
- Drag the mechanism through the desired range of motion.
- Analyze the velocity and acceleration of components in the assembly.
- Run the mechanism through its motion cycle.
- Create trace curves for moving parts in the assembly.
- Create a motion envelope for the moving parts in the assembly.
- Create an MPEG or other video of the mechanism in motion.

The tutorials in this textbook include many different mechanism designs as listed below. See Appendix A for reference drawings of each mechanism and the part names used in each design.

- Simplified 4 cylinder overhead cam engine as shown.
- Geneva Wheel index device.
- Scotch Yoke index device.
- Toy airplane and toy truck.
- Home garage door and opener.
- Rack and pinion device.
- Front end loader.
- Assembly machine mechanisms (3).
Configuration for Mechanism Design

The following config.pro options apply to Mechanism Design:

- **motion_envlp_alert**: Displays the motion envelope alert the first time the quality level is increased.
- **sim_display_motion_cams**: Toggles the display of cam connection symbols.
- **sim_display_motion_connections**: Toggles the display of connection symbols.
- **sim_display_motion_drivers**: Toggles the display of servo motor symbols.
- **sim_display_motion_gears**: Toggles the display of gear connection symbols.
- **sim_display_motion_ground_pts**: Toggles the display of ground point symbols.
- **sim_display_motion_lcs**: Toggles the display of local coordinate system symbols.
- **sim_display_motion_slots**: Toggles the display of slot connection symbols.
- **sim_motion_analysis_accuracy**: Specifies the default accuracy for motion analyses.
- **sim_motion_analysis_assem_tol**: Specifies the default error allowed for assembly analyses.
- **sim_motion_analysis_duration**: Specifies the default duration time for motion analyses.
- **sim_motion_analysis_increment**: Specifies the default increment between steps of motion analyses.
- **sim_motion_analysis_integrator**: Controls the default integrator used when performing a motion analysis.
- **sim_motion_analysis_method**: Specifies the default method used when performing a motion analysis.
- **sim_motion_analysis_start_time**: Specifies the default start time for motion analyses.
- **sim_motion_analysis_vel_tol**: Specifies the default error allowed when performing a velocity analysis.
- **sim_motion_output_fly_file**: Controls the creation of .fra file, which is used by Pro/FLYTHROUGH.
**Definitions and Terminology**

The following definitions and terms apply to Pro/ENGINEER *Mechanism Design*.

**Analysis**  
A study of the motion in the mechanism.

**Body**  
A component (part or sub-assembly) in the mechanism.

**Cam Connection**  
A special relationship between two bodies in the mechanism where one is a cam and the other is a follower.

**Connection**  
A special type of assembly constraint which allows under-constrained placement in the assembly model.

**Degrees of Freedom**  
The allowed motion of a body in the mechanism. Connections reduce the number of degrees of freedom for a given body.

**Drag**  
Use the mouse to move the mechanism through the range of motion.

**Dynamics**  
A study of the mechanism’s motion with regard to forces and loads. This textbook does not cover the *Mechanism Dynamics* module of Pro/ENGINEER Wildfire.

**Gear Pair Connection**  
A special relationship between two bodies in the mechanism where the velocity of one body is driven by the velocity of the other.

**Ground**  
A body that is fully constrained in the assembly, it has no degrees of freedom, it can not move.

**Joint**  
A special type of connection between two bodies in the mechanism.

**Kinematics**  
A study of the mechanism’s motion without regard to forces and loads.

**Local CSYS**  
The local coordinate system associated with a body, usually the default coordinate system in the part or sub-assembly.

**Motion**  
Movement of a body caused by motors or loads.

**Placement Constraint**  
The method used in standard Pro/ENGINEER assemblies to assemble components. Each constraint limits motion of the component in one direction.

**Playback**  
Record and replay the motion generated in a motion analysis.
Definitions and Terminology (continued)

Servo Motor
Define motion of a joint connection using velocity, acceleration, or position between two bodies.

Slot Connection
A special relationship between two bodies in the mechanism where one is a slot and the other is a follower.

User CSYS
Additional coordinate systems created in the mechanism.

World CSYS
The global coordinate system for the entire mechanism.

Mechanism Connection Symbols

Pro/ENGINEER Wildfire Mechanism Design uses a variety of symbols (sometimes called icons) to indicate Connections in the assembly. The most commonly used of these connection symbols are shown below.

- Cam
- Planar
- Cylinder
- Servo Motor
- Gear
- Slider
- Pin
- Slot
Joint Connections

Introduction

The Mechanism Design module uses special assembly constraints, called Connections. These allow the bodies to move in the mechanism assembly.

There are many different types of connections, and each one requires different references in the component and the assembly. Each connection type allows some type of movement in the assembly. For example, a Pin connection allows rotation of the component in the assembly.

Creating a Connection

To create a joint connection, pick Insert, Component, Assemble or pick the icon shown above. In the Component Placement dialog box, expand the Connections section, then select the type of connection as shown on the next page.

After selecting the type of connection, pick the appropriate references for the connection in both the component and the assembly. For some connection types, both references must be in the same two components or bodies of the assembly. For some connections, the Flip option is used to flip the body by 180°.

For some bodies, multiple connections may be required to fully connect the mechanism. Pick the ‘plus’ sign in the Component Placement dialog box to add additional connections.

Note

For some connection types, both references must be in the same two components or bodies of the assembly.
Creating a Connection (continued)

The Component Placement dialog box is shown below.

Expand the Connections section of the dialog box by picking here.

Set the Type of connection here.

Pick Flip to flip the body 180°.

Pick here to select the reference in the body being assembled.

Pick here to select the reference in the body being assembled to.

Pick OK to complete the connection.
Joint Connection Types

There are many types of joint connections including:

- Pin
- Slider
- Cylinder
- Planar
- Ball
- Weld
- Bearing
- Rigid

**Pin**

Select an axis (or revolved surface) and a plane in the component and the assembly. Both sets of references must be to the same two components in the assembly. This connection type allows rotational motion only as shown below.

![Pin Connection](image)

**Slider**

Select an axis (or edge) and a plane in the component and in the assembly. Both sets of references must be to the same two components in the assembly. This connection type allows linear motion only as shown below.

![Slider Connection](image)
Planar
Select a plane in the component and in the assembly. This connection type allows linear and rotational movement within the selected plane as shown below, and is used in conjunction with other connections.

Cylinder
Select an axis (or revolved surface) in the component and in the assembly. This connection type allows both linear motion along the axis and rotational motion about the axis as shown below.
Task 10: Add more joint connections.

- **Hide** the following components of the assembly:
  - oil_pan.prt
  - engine_block.prt
- Assemble the part called crank_shaft
- Expand the **Connections** section of the dialog box here
- Set the connection type to **Pin** here
- Pick these two axes
- Pick the FRONT datum plane in the crankshaft and pick the FRONT datum plane in the lower block
- Be sure that both the axis reference and the translation reference are to the lower block part and not to the engine assembly
• Using CTRL, ALT and the middle mouse button, spin the crankshaft to the approximate rotation as shown below (this just makes it easier later)

• Pick **OK** to complete the connection

• The result is shown here

• **Hide** the part called lower_block

• Assemble the part called connect_rod

• Expand the **Connections** section of the dialog box

• Be sure the connection type is set to **Pin**

• Pick the two axes as shown below
• Pick the FRONT datum plane in the connecting rod and pick the DTM1 datum plane in the crankshaft
• Be sure that both the axis reference and the translation reference are to the crankshaft part and not to the engine assembly
• Pick the Plus sign to add another connection
• Set the connection type to Cylinder
• Pick the two cylindrical surfaces as shown below


**Introduction**

The Mechanism Design module allows the creation of Trace Curves and Motion Envelopes for the mechanism. Each of these uses existing analysis result sets to calculate the geometry.

**Trace Curves**

Trace Curves are created by the system to indicate the path traveled by a selected point or vertex during a Kinematic analysis of the mechanism.

Trace Curves are created in a special part that you must define in the mechanism, called the ‘Paper’ part. The best method is to create an empty part in the assembly to be used for this purpose. Pick Insert, Component, Create and add the part using the Default constraint.

To create a Trace Curve, enter the Mechanism module, then pick the Trace Curve icon, shown above. Pick the ‘paper’ part, then select a datum point or vertex whose path you want to trace. Examples of Trace Curves are shown below.
Trace Curves (continued)

The Trace Curve dialog box is shown below.

Creating Trace Curves can generate several errors as shown below.
Using Trace Curves

Trace Curves can be used to create a cam profile, a slot curve, and can be used to create other geometry in standard Pro/ENGINEER. The Trace Curve is created as a feature in the ‘paper’ part as shown below.

The Trace Curve is created using a motion analysis and a selected vertex

The ‘Paper’ part is a separate part in the assembly