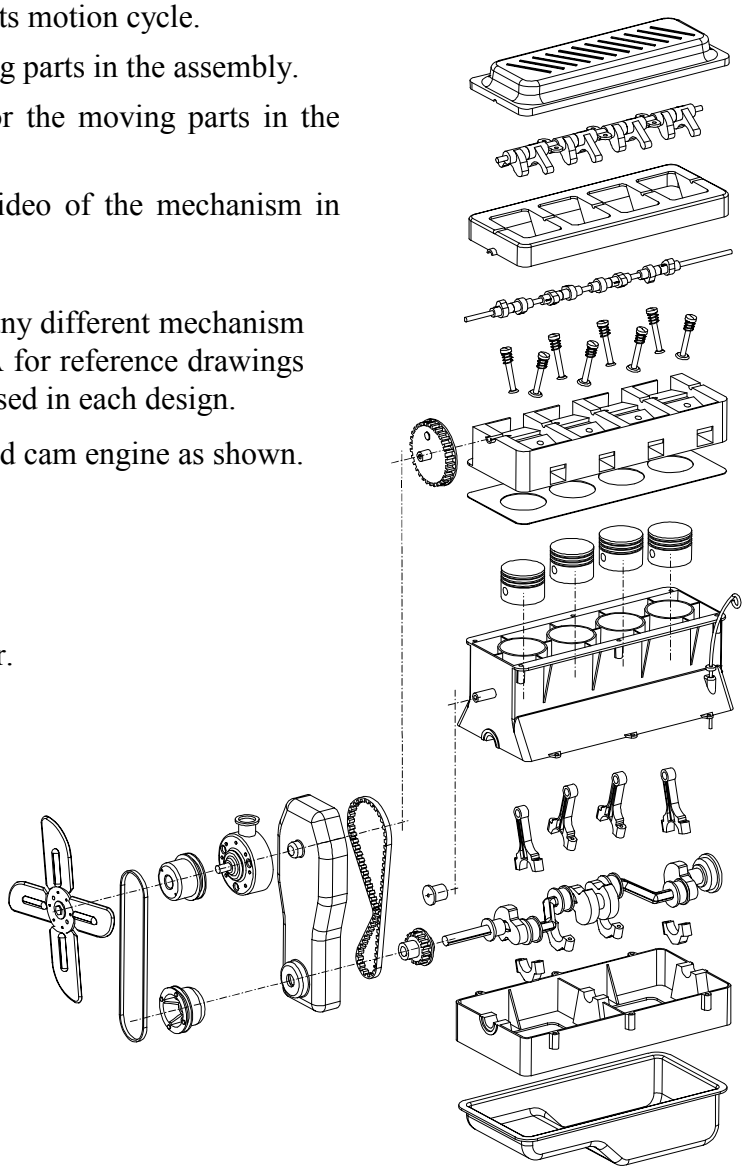
Introduction to Mechanism Design

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**.

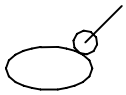
Analysis	A study of the motion in the mechanism.
Body	A component (part or sub-assembly) in the mechanism.
Cam Connection	A special relationship between two bodies in the mechanism where one is a cam and the other is a follower.
Connection	A special type of assembly constraint which allows under-constrained 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)

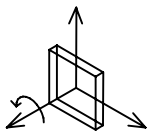
Servo Motor	Define motion of a joint connection using velocity, acceleration, or position between two bodies.
Slot Connection	A special relationship between two bodies in the mechanism where one is a slot and the other is a follower.
User CSYS	Additional coordinate systems created in the mechanism.
World 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.



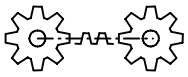
Cam



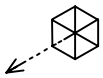
Planar

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°.

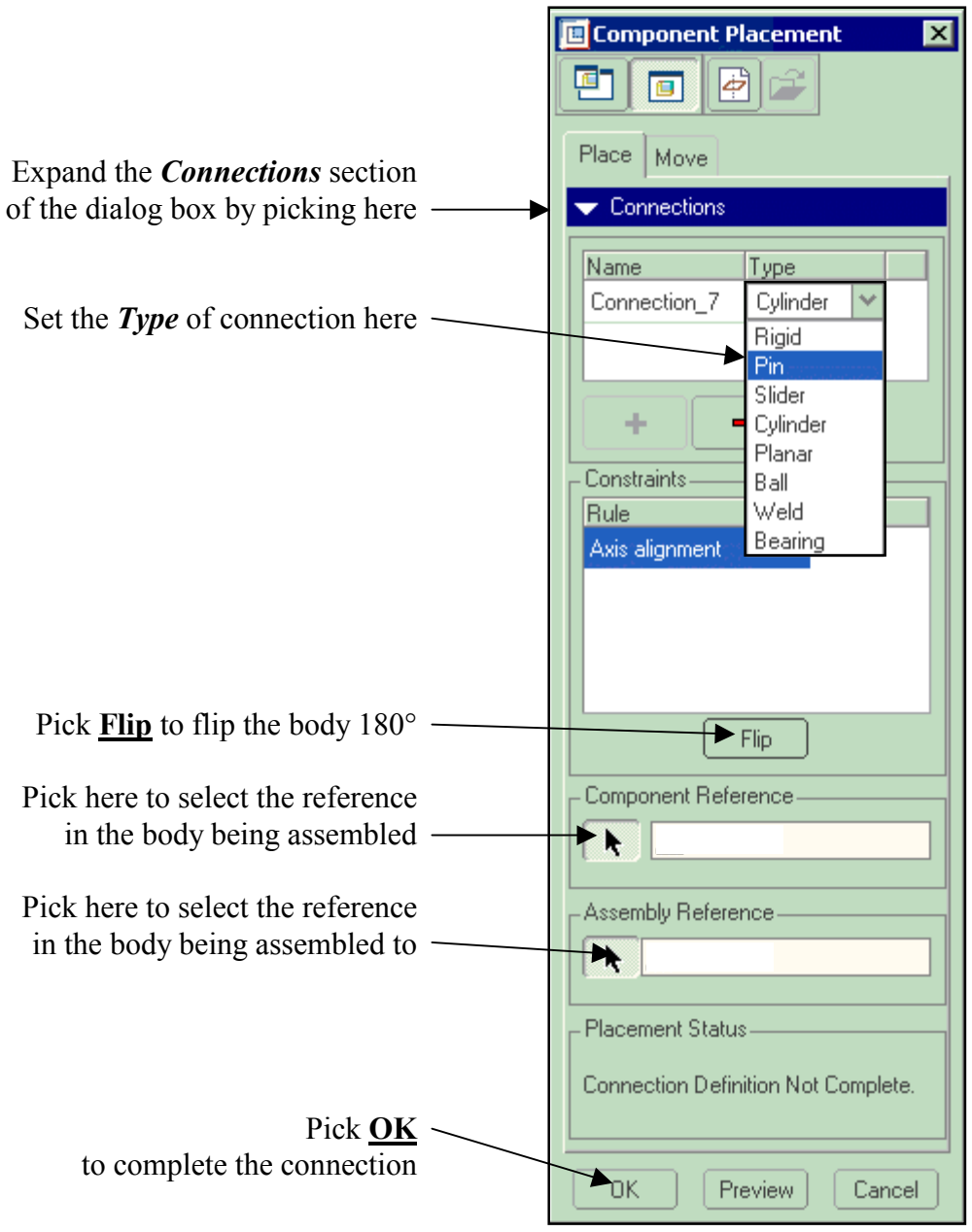
For some bodies, multiple connections may be required to fully connect the mechanism. Pick the 'plus' sign in the **Component Placement** dialog box to add additional connections.

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.



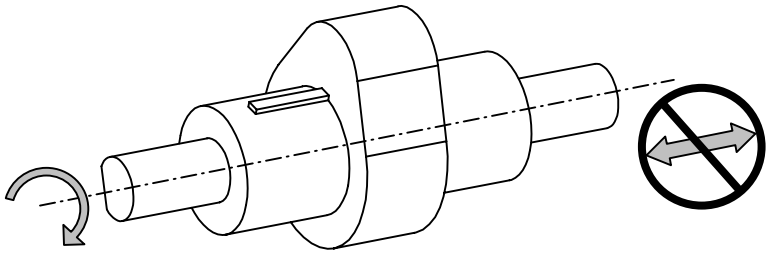
Joint Connection Types

There are many types of joint connections including:

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

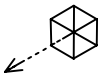
Pin

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.

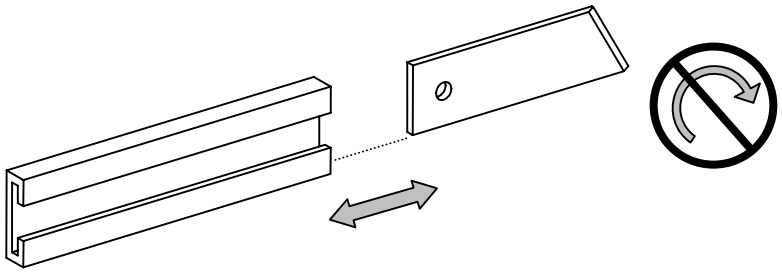


Pin Connection

Slider

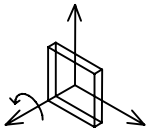


Select an axis (or edge) and a plane in the component and in the assembly. Both sets of references must be to the same two components in the assembly. This connection type allows linear motion only as shown below.

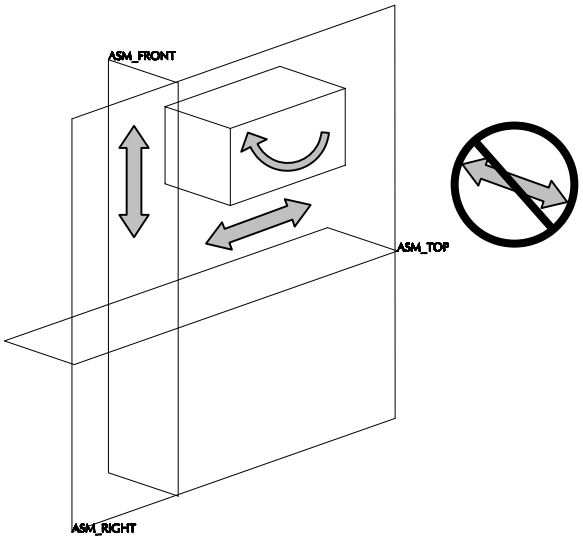


Slider Connection

Planar



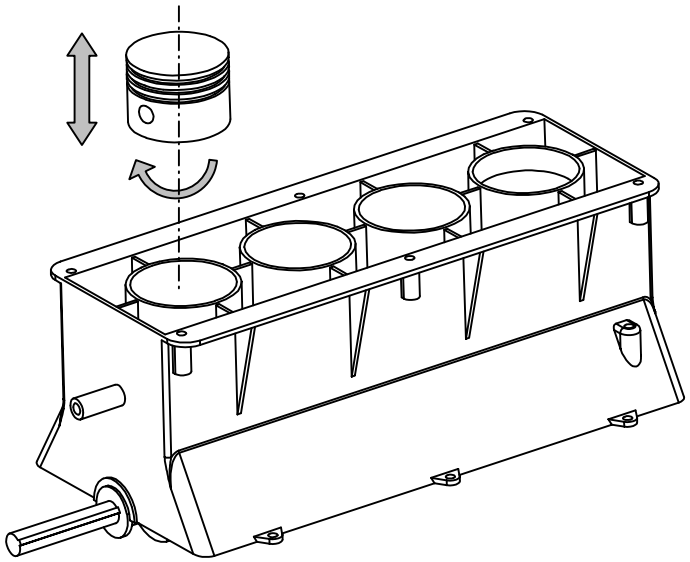
Select a plane in the component and in the assembly. This connection type allows linear and rotational movement within the selected plane as shown below, and is used in conjunction with other connections.



Planar Connection

Cylinder

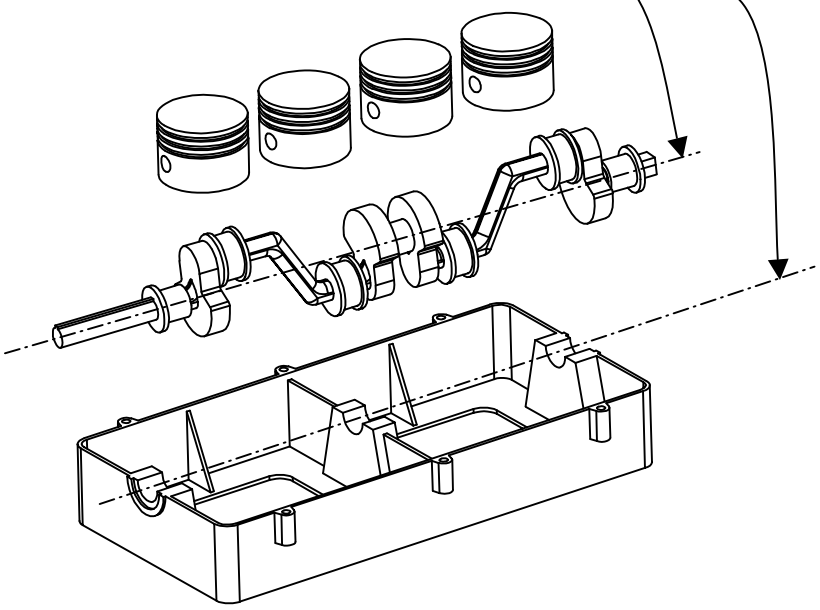
Select an axis (or revolved surface) in the component and in the assembly. This connection type allows both linear motion along the axis and rotational motion about the axis as shown below.

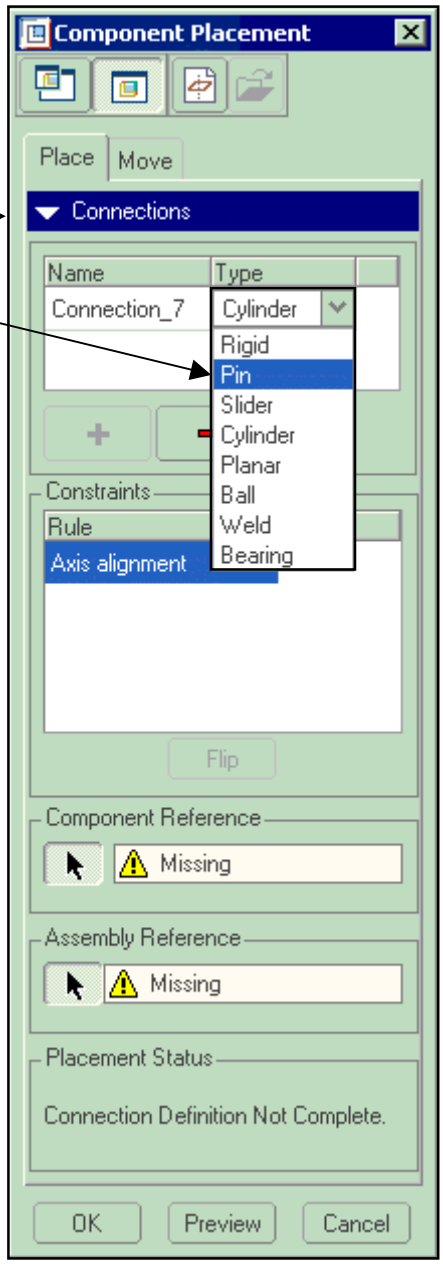


Cylinder Connection

Task 10: Add more joint connections.

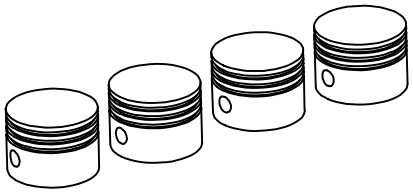
- **Hide** the following components of the assembly:
  - oil_pan.prt
  - engine_block.prt
- Assemble the part called crank_shaft
- Expand the **Connections** section of the dialog box here
- Set the connection type to **Pin** here
- Pick these two axes

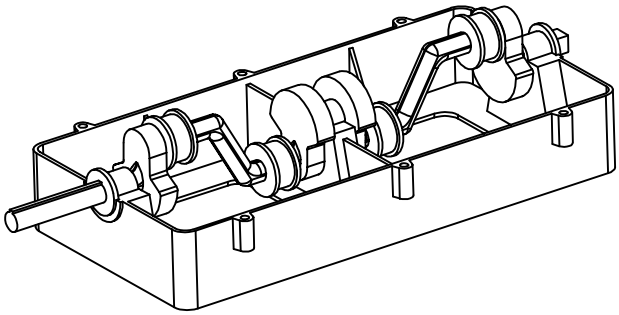




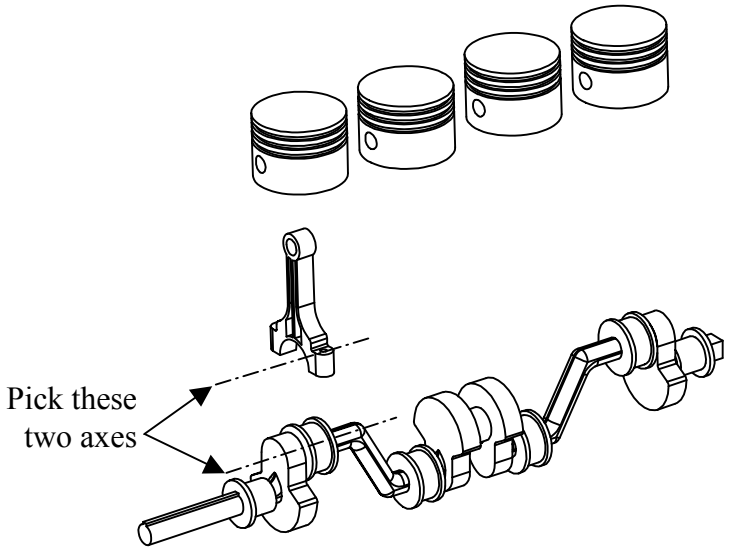
- Pick the FRONT datum plane in the crankshaft and pick the FRONT datum plane in the lower block
- Be sure that both the axis reference and the translation reference are to the lower block part and not to the engine assembly

- 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

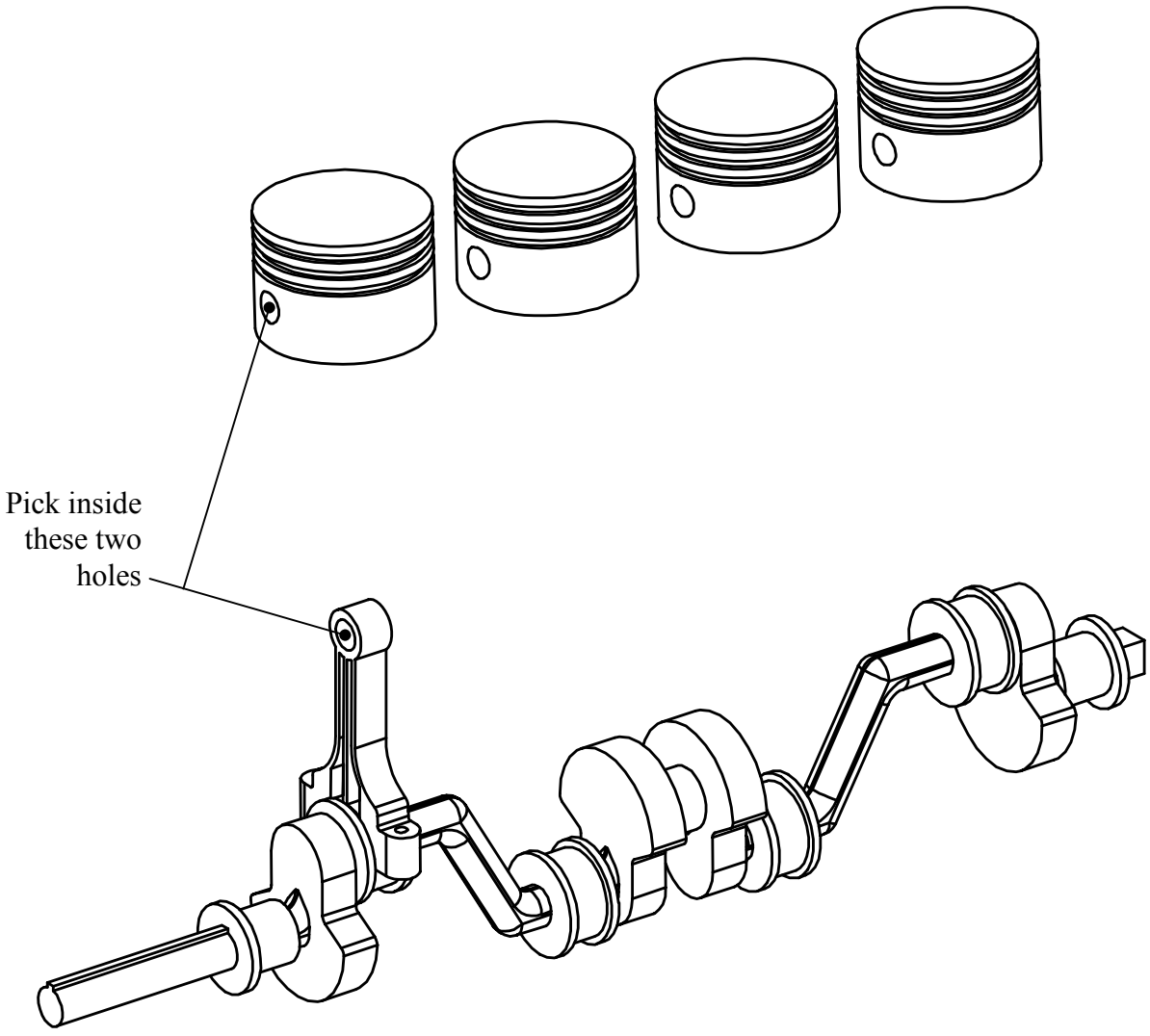




- **Hide** the part called lower_block
- Assemble the part called connect_rod
- Expand the **Connections** section of the dialog box
- Be sure the connection type is set to **Pin**
- Pick the two axes as shown below



- 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



Trace Curves and Motion Envelopes

Introduction

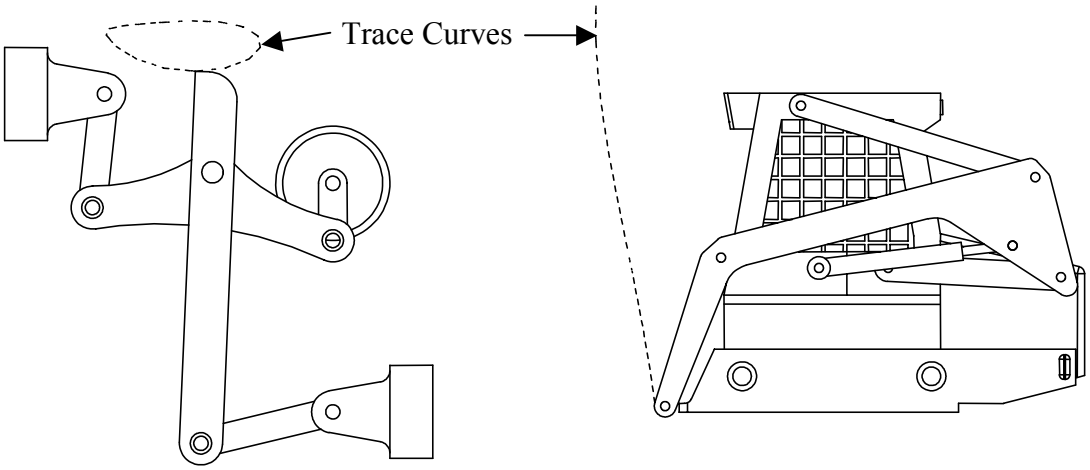
The **Mechanism Design** module allows the creation of **Trace Curves** and **Motion Envelopes** for the mechanism. Each of these uses existing analysis result set to calculate the geometry.

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.

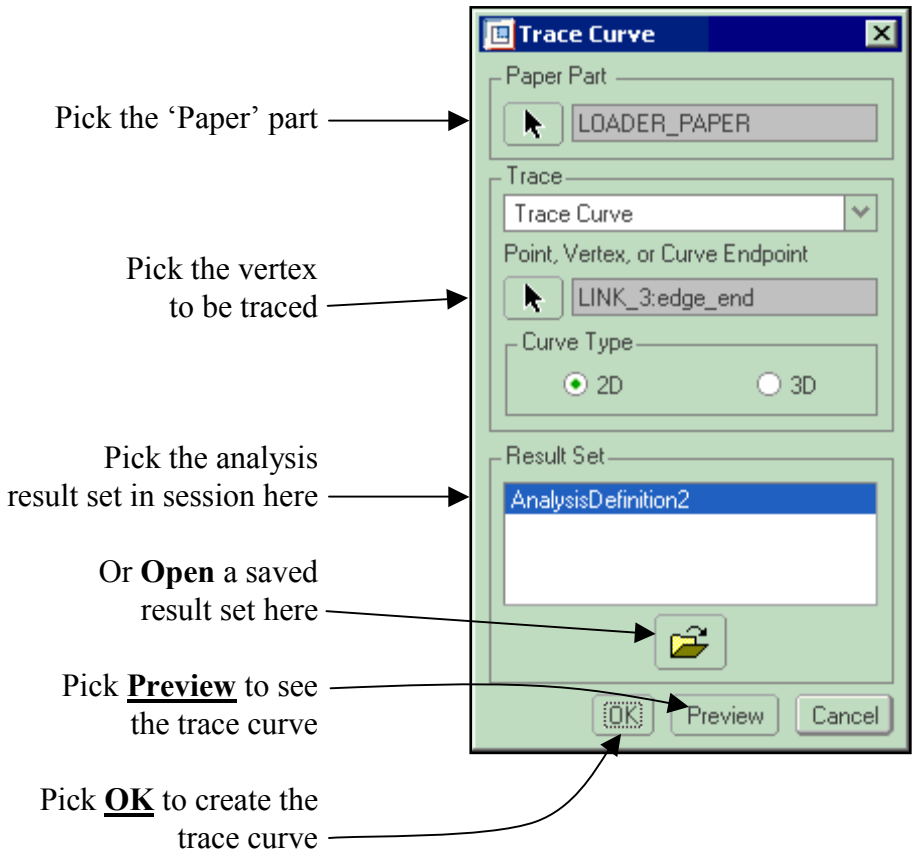
Trace Curves are created in a special part that you must define in the mechanism, called the 'Paper' part. The best method is to create an empty part in the assembly to be used for this purpose. Pick **Insert, Component, Create** and add the part using the **Default** constraint.

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.

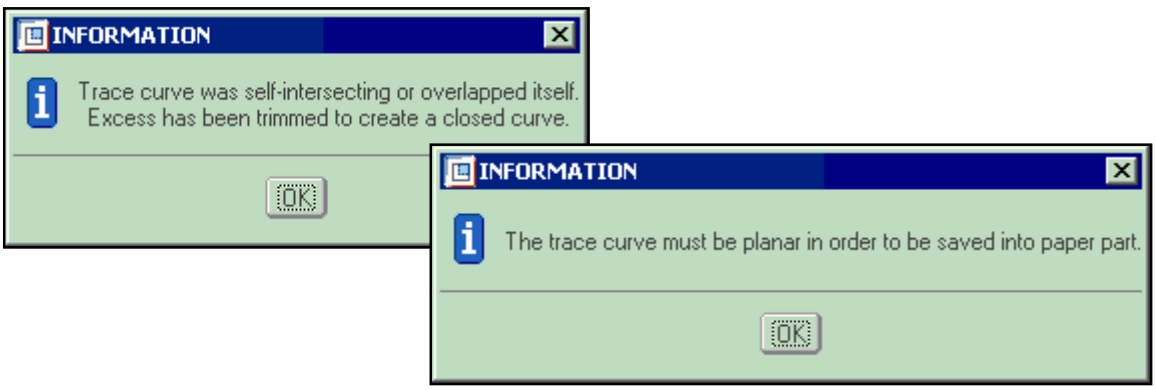


Trace Curves (continued)

The **Trace Curve** dialog box is shown below.



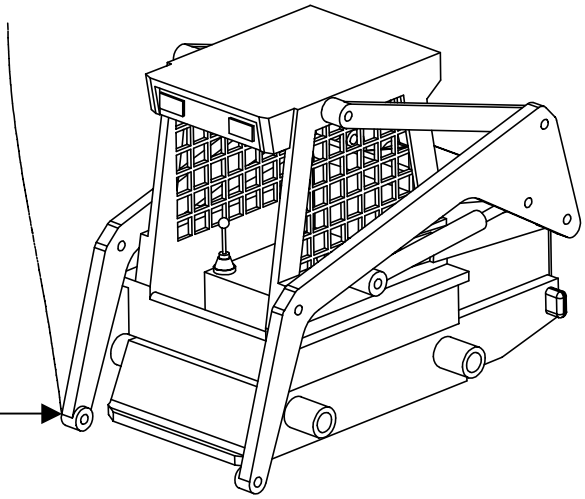
Creating **Trace Curves** can generate several errors as shown below.

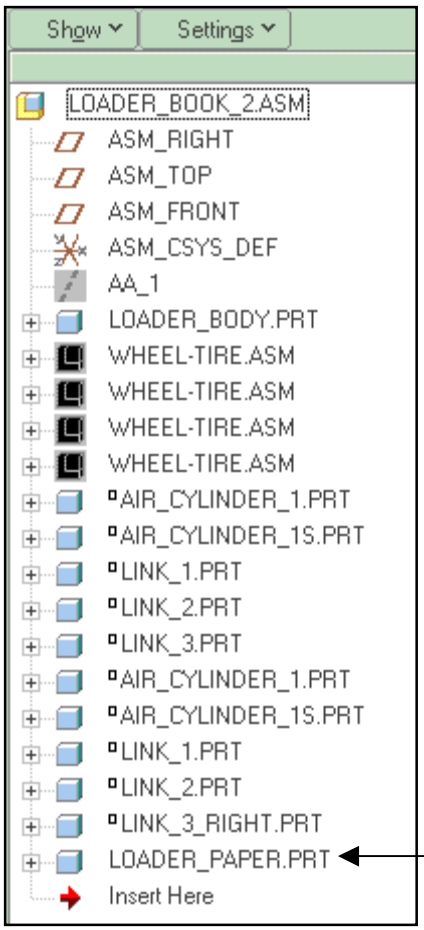


Using Trace Curves

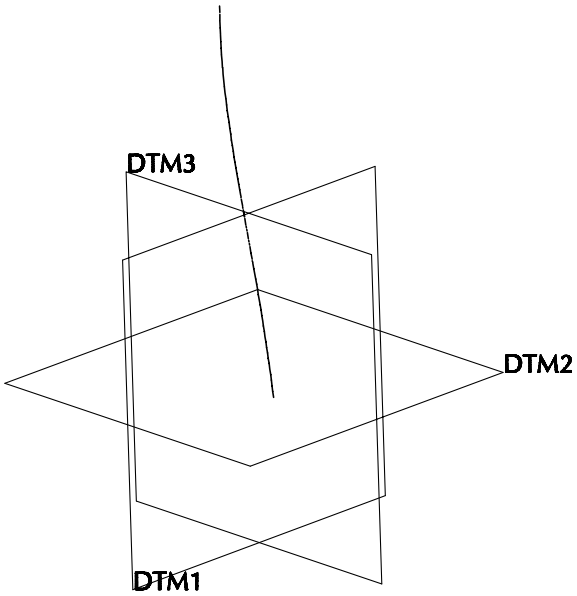
Trace Curves can be used to create a cam profile, a slot curve, and can be used to create other geometry in standard Pro/ENGINEER. The **Trace Curve** is created as a feature in the 'paper' part as shown below.

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





The 'Paper' part is a separate part in the assembly



The Trace Curve is created in the 'Paper' part