1. Introduction & Installation

Working Model - Fast and Powerful Engineering Analysis for Desktop PC's

Working Model is a powerful tool for engineering analysis, animation, and prototyping. It saves time and money in the design process by allowing you to create and analyze dynamic physical systems on the computer prior to building costly prototypes.

Operating Concept

The operating concept of Working Model is straight-forward. First, define a set of rigid bodies and constraints (e.g., motors, springs, and joints) by drawing them with a mouse. Then set your system into motion by selecting "RUN" -- there is no pre-processing or post-processing. You immediately receive accurate results.

With Working Model, you create systems that are driven by physical laws. You can test, re-design, and re-test your mechanical system, speeding your time to market by seeing which designs work better before you build them.

Working Model allows you to fine-tune simulation parameters. You can define controllers to adjust properties of objects. You can create meters to plot the data that is taken during a simulation. You can design a model in your CAD program and import the data into Working Model. You can even use another application such as Excel or MATLAB® to control your simulations.

Designed with Ease -of-Use in Mind

Working Model was designed to be an integral tool in the design and analysis process for engineers of all types. Its highly intuitive interface makes it useful for engineers of all levels. An engineer can quickly test the performance of a shock absorber with a simple model, or create a highly complex dynamic model of an automobile engine. With its high degree of accuracy, Working Model can simulate almost any mechanical system.

Parametrics

Use equations to define the relationships among components of your design. When you re-position or re-size components, your design maintains the connections, sizing, and positions specified by your equations.

Smart Editor ™

Working Model features the unique Smart Editor™. The Smart Editor shortens the design process by making it extremely easy to build and maintain complex designs as well as check for clearances, mechanism functionality, and tolerances. Interconnected objects can be moved by simply clicking and dragging—the Smart Editor automatically moves each object based on existing constraints.

Import / Export

Working Model includes import/export options. With Working Model, users can import CAD drawings in DXF format from popular applications such as AutoCAD®, CADKEY and Vellum® and use them immediately in simulations.

Inter-Application Communication

Working Model uses Dynamic Data Exchange (DDE) on Windows and Apple® Events on Macintosh to communicate with other applications during a simulation. With this feature, users can specify physical models of real-life mechanical designs and then control them externally through other programs. For example, Microsoft® Excel can be used to calculate control signals. Data from Working Model is sent to the spreadsheet which calculates the control signals based on the current state of the system. New control information is then received by Working Model and used to calculate the next simulation frame.

About the Demonstration Version

These demonstration disks include a limited version of Working Model along with numerous example files. You will be able to build parametric models, simulate their behavior, make measurements and interact with systems using the Smart Editor™.

Several features, however, have been disabled in the demonstration version. In particular, file saving functionality has been removed (Save, Save As, Print, Export, Cut, Copy, Paste and Duplicate). This version also limits the number of measurements and bodies that can be created to one meter and five bodies per document.

The Demonstration Guide & Tutorial

This guide will provide you with information you will need to evaluate Working Model. It contains a 10 Minute Demo, two tutorials, and reference information on Working Model's tool palette, function language, and parametrics.

System Requirements

Working Model versions are available for both Macintosh and Windows personal computers. The following system configurations are needed to run Working Model:

Windows: A 486 microprocessor or better is required, Windows version 3.1 or greater, 16 megabytes of RAM is strongly recommended (8 megabytes of RAM will require virtual memory and will slow simulations considerably) and 12 megabytes of available hard disk space.

Macintosh: A 68020-based computer or higher (Mac II or above) is required. Macintosh Plus, SE, Portable, Classic and Powerbook 100 are not supported. Working Model requires System 7 or later, approximately 10 megabytes of RAM and 8 megabytes of available hard drive space. A floating-point math co-processor is highly recommended. Working Model is compatible with PowerPC-based Macintosh computers.

Installation

The demonstration disk contains an automatic installer. To install:

On Macintosh: simply double-click on the icon named **Double-Click Me to Install**.

On Windows: run **SETUP** from within Windows.

Ordering Information

We hope you enjoy the demonstration version. To order your full copy of Working Model:

- contact your local authorized reseller,
- call Knowledge Revolution TOLL-FREE at **(800) 766-6615** or **(415) 574-7777** (8:00AM to 5:00PM Pacific Time),
- e-mail us (internet) via *info@krev.com*, or

- print out the order form in the **README** file included on the demonstration disks and fax or mail it to Knowledge Revolution at:

FAX: **(415) 574-7541**

MAIL: **Knowledge Revolution**
66 Bovet Road, Suite 200
San Mateo, CA 94402

Working Model comes with a 30-day money-back guarantee!

2. The Ten Minute Demo

Working Model's graphical users interface allows users to model and analyze systems very rapidly. This 10 minute demo walks you through two simple examples - a ball bouncing down a ramp, and a 4-bar linkage.

A Ball Bouncing Down a Ramp

Creating simulations in Working Model is similar to drawing in any popular CAD package. Using the tools found in the toolbar, you first sketch bodies, then connect them together using Working Model's many different constraints.

When all bodies and constraints are defined, the simulation is run. No pre-processing or post-processing is needed. You immediately see results.

Step 1 - Open Working Model

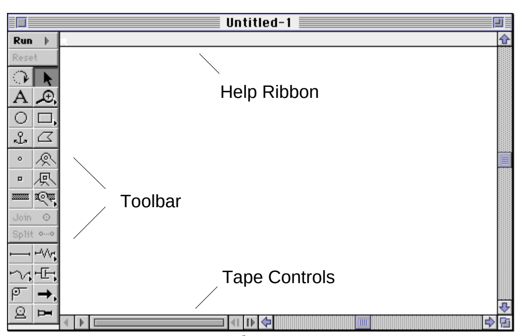
1. **Double-click the Working Model icon to start the demo program.**

   Working Model starts up and begins running a demo script written in WM Basic,Working Model's embedded scripting language.

2. **A dialog box will pop up, offering you several options.**

** Change picture.

Figure 2-1
Untitled Working Model Window



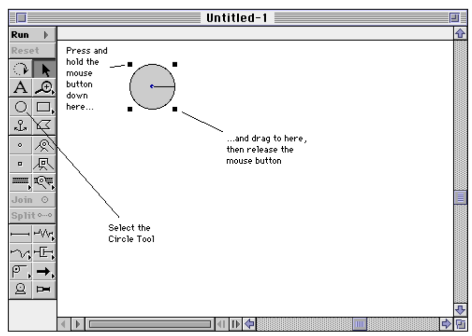
You will see the toolbar on the left and the tape controls along the bottom of the window. The toolbar contains tools you will use to create simulations. The tape controls give you more control for running and viewing simulations. You can use the tape controls to step through simulations, play simulations backwards, or move to a specific time in a simulation.

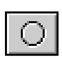
Step 2 - Draw a circle

The toolbar provides a variety of tools for setting up simulations. To choose a tool, click on the corresponding icon on the toolbar. The Help Ribbon located at the top of the window helps you identify the tools as you move the mouse pointer over the tool icons.

To draw a circle (see Figure 2-2):

Figure 2-2
Drawing a circle





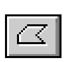
1. **Click the Circle tool.**
2. **Position the pointer at any starting point in the blank area of the screen.**

   The pointer changes from an arrow to a crosshair. This means you are ready to create an object.

3. **Press and hold down the mouse button (left button on a two-button mouse). Drag the mouse to draw a circle. When the circle reaches the size you want, release the mouse button.**

   A line appears inside the circle. During an animated sequence, this line indicates the circle's rotation.

Step 3 - Draw a Ramp

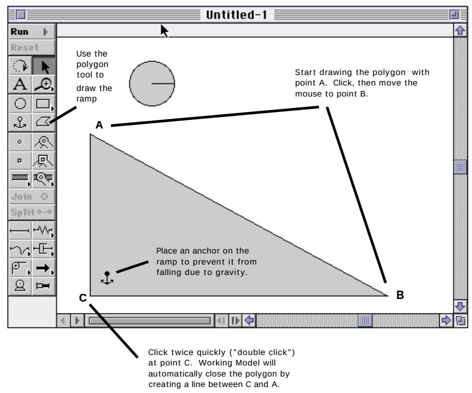


1. **Click the polygon tool.**

   The pointer changes from an arrow to a crosshair. This means you are ready to create an object.

2. **Position the pointer to draw the upper left point (point A) of the polygon as shown in Figure 2-3.**

Figure 2-3
Creating a ramp with the polygon tool



3. **Click the mouse button (left button on a two-button mouse) and release. Move the mouse to the lower right point (point B) of the polygon.**

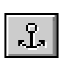
   Refer to Figure 2-3. A line is drawn as the polygon is being created.

4. **Click the mouse button again and release. Move the mouse to the bottom left point (point C) of the polygon.**

   Refer to Figure 2-3. A second line is drawn as the polygon is being created.

5. **Double click (click twice quickly) on the mouse button.**

   The polygon is automatically closed by Working Model, creating a ramp.



6. **Click the anchor tool.**

7. **Position the anchor over the ramp with the mouse and click.**

   The anchor pins the ramp to the background, preventing gravity from dragging it downward.

Step 4 - Run The Simulation

You are now ready to run your simulation. To run the simulation:

1. **Click Run in the toolbar.**

Watch your first simulation run. Because normal earth gravity is on by default in a new document, the circle drops and bounces down the ramp, with Working Model automatically handling all contacts and collisions. Note that Working Model also handles friction -- the ball spins as it rolls down the ramp.

2. **Click once in an empty area of the workspace to stop the simulation.**

Step 5 - Display a velocity meter

Working Model allows you to measure many physical properties such as velocity, acceleration, and energy by using meters and vectors. Meters and vectors provide visual representations of quantities you want to measure.

To display a digital meter that measures the velocity of the projectile, follow these steps:

1. **Click Reset in the toolbar.**
2. **Select the circle drawn earlier by clicking on it.**
3. **Choose Velocity from the Measure menu at the top of the screen, and choose All from the Velocity sub menu (see Figure 2-4).**

   A digital velocity meter appears (Figure 2-5).

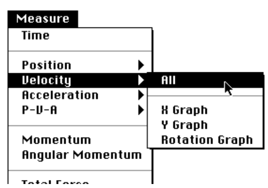


Figure 2-4
Choosing All from the Velocity submenu in the Measure menu

4. **Click Run in the toolbar.**

   See Figure 2-5. Tracking has also been turned on (by selecting Tracking, Every 8 Frames, under the World menu). Notice that Working Model handles contacts, collisions and frictions automatically.

Step 6 - Change simulation parameters

Every object in Working Model, even input controls, has a series of properties, like mass, material, position, etc. These properties can be viewed and modified from the properties window.

Let’s change the material of the circle to rubber.

1. **Double-click on the circle.**

   The properties window will appear, allowing you to view the circle’s position, angle, velocity, material, mass, friction, elasticity, charge, and moment of inertia. Any of these properties can be edited directly within the properties window.

2. **Position the pointer over the arrow on the “material” drop-down menu.**

3. **Press and hold down the mouse button (left button on a two-button mouse). Drag to the material “rubber” and release the mouse button.**

   You can edit other properties by clicking in the box you wnat to change and typing in a new value.

4. **Click Run in the toolbar.**

5. **Modify other parameters and re-run.**

   Double-click again on the ball to open its properties window and change its mass, elasticity, and friction coefficient. Re-run the simulation. Similarly, change the properties of the ramp. Add other objects above the ramp and simulate multi-body collisions.

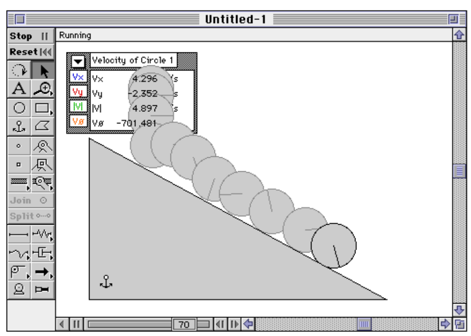


Figure 2-5
Running the simulation with tracking and a meter

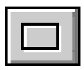
Modeling a Four Bar Linkage

Creating mechanisms with Working Model is a snap. In this example, you will create a parametrically-defined four-bar linkage using the Smart Editor.

Step 1 - Create the bars

To construct a linkage consisting of three bars:

1. **Close all open documents prior to starting this demo.**
2. **Create a new Working Model document by selecting New from the File menu.**



3. **Double click the rectangle tool on the toolbar.**

   The tool will turn black, indicating that it can be used multiple times.

4. **Sketch a rectangle similar to the one in Figure 2-6.**

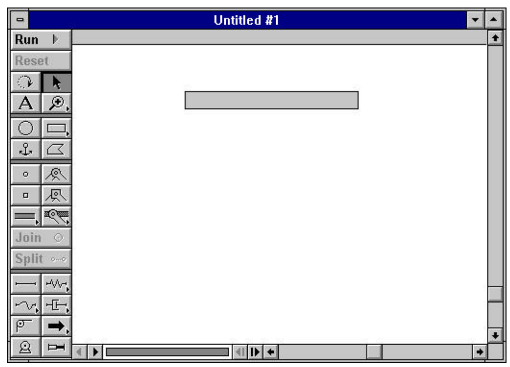


Figure 2-6
A single rectangle

5. **Sketch two vertical rectangles below the horizontal rectangle.**

   *None of the rectangles should overlap. **Fix the picture bewlow.***

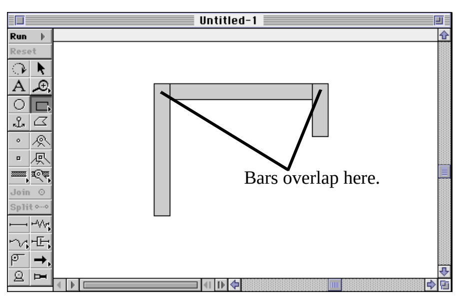
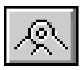


Figure 2-7
The beginnings of a four-bar linkage

Step 2 - Placing points on the bars

Here you will learn to use the Object Snap tool to place points precisely. The Object Snap tool highlights commonly-used positions, like the center of a side, with an “X”. If you place a point using Object Snap, the point's position will be marked by parametric equations. These equations insure that the point maintains its relative location even after re-sizing or other adjustments.



1. **Double click on the Point tool. **Change the icon at left.****

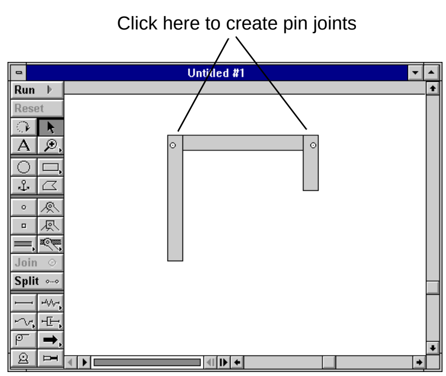
   The tool will turn black, indicating that it can be used multiple times.

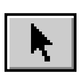
2. **Move the cursor over one of the bars.**

   You will notice that an "X" appears around the pointer when it is centered on a side, over a corner, or over the center of a rectangle. This feature is called "Object Snap" and highlights the commonly-used parts of a body.

3. **Place the cursor over the center of the top side of one of the vertical bars. When you see an "X" around the pointer, click the mouse button. ** Change picture below.**

The beginnings of a four-bar linkage





4. **Place points, as you did in step 3, like shown in figure ???. Make sure you see the "X" when you click the mouse.**

5. **Selet the Pointer tool.**

6. **Double-click on one of the points to bring up the properties window.**

   You will notice that the point is defined locally by 0.0 in one direction -- so the point stays in the center of that side -- and by an equation for the other direction -- half the length or width of the body -- so the point always stays at the very end of the body. You may have to re-size the properties window to see the full equation.

Step 3 - Connecting the points to form pin joints

You will now join the points to create pin joints. A pin joint acts as a hinge between two mass objects. The Smart Editor will prevent joints from breaking during a drag operation.

1. **With the pointer tool selected, click and drag on the background to make a selection box that surrounds the two left points. Release the mouse button, and the two points should now be highlighted (darkened).**

   This method of selecting objects is called "box select." Any object that rests completely within the box when the mouse is released will be highlighted.

2. **Now the Join button in the toolbar will be highlighted. Click it.**

   The two points come together to form a pin joint. The bars may not be straight anymore, but we'll fix that in a minute.

3. **Perform steps 1 and 2 for the two right points.**

4. **Select the Pin Joint tool (show icon) and place a pin joint, again using snap points, at point B as indicated in Figure 2-8.**

   This pin joint will join the rectangle to the background.

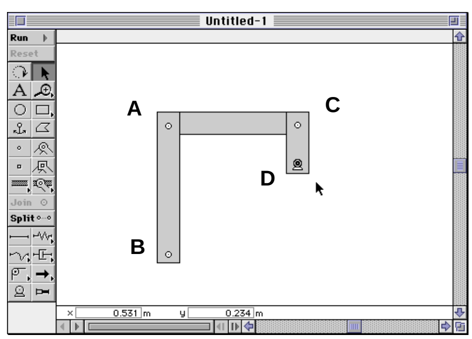


Figure 2-8
Adding a motor to the linkage.

Step 4 - Adding a Motor to the Linkage

We will now attach a motor to one of the bars to drive the linkage.

1. **Click the motor tool on the toolbox.**

   The motor tool will become shaded, indicating that it has been selected. The cursor should now look like a small motor.

2. **Place the cursor over the bottom of rectangle D (see Figure 2-8). Click the mouse.**

   A motor will appear on the 4-bar linkage as shown in Figure 2-8. Similar to a pin joint, a motor has two attachment points. A motor will automatically connect the top two mass objects. If only one mass object lies beneath the motor, it will join the mass object to the background. A motor will then apply a torque between the two mass objects it is pinned to.

3. **Click Run on the toolbar.**

   The 4-bar linkage will begin slowly cranking through its range of motion.

4. **Click Reset in the toolbar.**

   The simulation will stop and reset to frame 0.

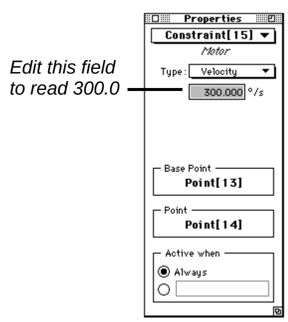
5. **Double-click on the motor to open the properties box.**

   You can also select the motor and choose Properties from the Window menu to open the properties box.

6. **Increase the velocity of the motor to 300 degrees/second by typing this value in the properties box as shown in Figure 2-9**

   Users can define a motor to apply a certain torque, to move to a given rotational position, or to turn at a given velocity or acceleration. Rotation, velocity, and acceleration motors have built-in control systems that automatically calculate the torque needed. In this demo, we will stick with the velocity motor.

Figure 2-9
Properties window for a motor



7. **Click Run on the toolbar.**

   The 4-bar linkage will once again begin cranking, this time at a much higher velocity.

Step 5 - Re-sizing the bars

You will now use the Edit Toolbar on the bottom of the screen to adjust the size and angle of the bars. This section highlights Working Model's parametric features. Notice that when you re-size a bar all points stay in their proper positions and all joints stay intact. Because you located the points using Object Snap, they are positioned with equations and automatically adjust during design changes.

1. **If not already selected, click the Pointer tool.**
2. **Click once on the left-hand bar to select it.**

   Four black boxes will appear around the bar indicating that it is selected. The boxes are called handles and can be dragged to re-size an object.

3. **Double-click in the "h" box on the Edit Toolbar at the bottom of the screen to change the bar's height. Type 2.**

   The bar will re-size on the screen.

4. **Double-click in the "w" box on the Edit Toolbar to change the bar's width. Type .5.**
5. **Double-click in the "ø" box on the Edit Toolbar to change the bar's angle. Type 0.**

   The bars probably shifted when you joined them. Setting the angle to 0 straightens them back out.

6. **Change the height, width, and angle of the horizontal bar to 1, .5, and 0, resepctively.**
7. **Change the height, width, and angle of the right-hand rectangle to .5, 2, and 0, respectively.**

   Notice that during this re-sizing the pin joints all stay centered on the rectangles. Parametric equations keep the points in their proper relative positions.

Step 6 - Using the SmartEditor

The SmartEditor allows you to drag a mechanism through its range of motion while keeping all joints and connections intact. You can check clearances or set new starting points for the running of your simulation.

1. **Click on the small bar and hold the mouse button down.**

2. **Now, still holding the mouse button down, drag the linkage.**

   All three rectangles will follow the motion of the mouse, because the pin joints connect them. The Smart Editor does not allow joints to separate.

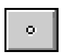
3. **Let go of the mouse button and click Run.**

   Then drag your design to a different spot and click run. You can use the SmartEditor to check the effect of different starting positions on your design.

Step 7 - Measuring a Point's Position

1. **Click Reset in the toolbar.**

   The simulation will stop and reset to frame 0.



2. **Select the point tool from the toolbar.**

   The point tool is located immediately to the left of the pin joint tool used earlier.

3. **Place the cursor over the horizontal bar of the 4-bar linkage and press the mouse button.**

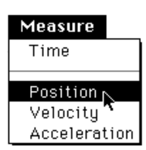
   A point is attached to the bar. This is a single point, and does not attach the bar to the background.

4. **If the point is not already selected (darkened), select it by clicking on it.**

5. **Create a meter to measure the position of this point by choosing Position from under the Measure menu (see Figure 2-10).**

   A new meter will appear. Position meters default to display digital (numerical) information. You can change a digital meter to a graph by clicking once on the triangle in the meter's upper left hand corner on Windows machines, or select the graph display from a pull down menu that appears when you click on the upper left hand corner of the meter in the Macintosh version.

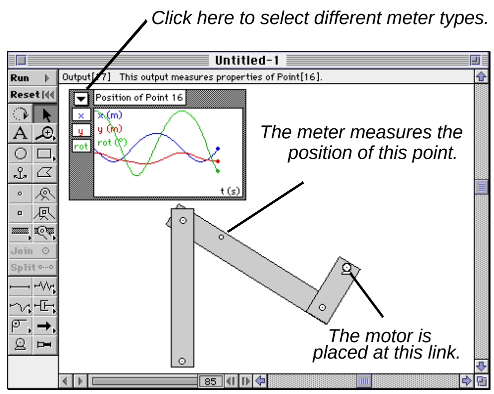
Figure 2-10
Creating a meter to measure a pins position



6. **Click Run on the toolbar.**

Your simulation will immediately begin running and measurement information will appear in the meter as shown in Figure 2-11. Meter data can be exported to an ascii file or copied onto the clipboard and pasted into a spreadsheet program for further analysis. In this case the spreadsheet would receive four columns of information: Time, X, Y, and Rotation. One row would appear for each integration time step calculated.

Figure 2-11
Running the simulation with a motor and a meter



7. **Modify your simulation and re-run.**

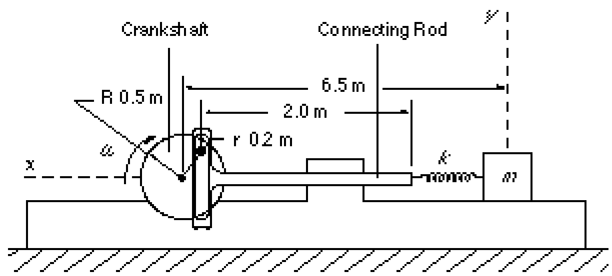
   Working Model's seamless integration between the editing of a model and the running of the dynamics engine allows you to quickly investigate many different simulation configurations. As an example, modify the mass of the horizontal beam using the properties box, and re-run the simulation. You can also modify pin joint locations, re-size bodies, measure velocities and forces, or even investigate how this 4-bar linkage would run in zero gravity by turning gravity off under the World menu!

Applying What You Have Learned

You have now finished the 10 minute demo. This demo has shown you how to create and run simple simulations in Working Model. We encourage you to experiment with the simulations you have already created, or create your own mechanism from scratch. As you will find, Working Model has an incredible array of features that will allow you to model and analyze most complex mechanical devices.

The following tutorials continue to explore the features in Working Model. For more information on the toolbar or Working Model's built in function language, please refer to the appendices at the end of this demo guide.

3. Tutorial: Driven Oscillation



This tutorial introduces you to several Working Model features:

- Slot Joints
- Springs
- Motors
- Custom graphs
- Slider controls

The problem solved with this tutorial is defined as follows:

Determine the rate (ω) in radians per second that the crankshaft must rotate so that resonance occurs. The crankshaft's radius R is 0.5 meters and the slider pin's position is a distance r = 0.2 meters from the crankshaft's center. The spring constant k is 50 N/m and the 0.6 meter square block's mass m is 1 kg. All other masses are negligible. Assume that there is no friction.

Introduction

As the frequency of the crankshaft approaches the natural frequency of the spring/mass combination, the amplitude approaches infinity. This condition is known as resonance. As we will demonstrate in this simulation, resonance can lead to catastrophic failure of structures and mechanisms.

You will use a motor and a circular mass object to model the driver crankshaft. The motor's speed will be controlled by an input slider. The connecting rod will be modeled as a rectangular mass object attached to a slot on the background.

Setting Up the Workspace

For this exercise four changes in the workspace are necessary. For clarity, the x-y axes will be displayed. The number of digits displayed after the decimal point will be changed to 1, from the default 3, to match the significant figures of the problem. Third, gravity will be turned off. Lastly, the automatic equation feature of Object Snap will be turned off.

1. **Choose Workspace from the View menu and select X,Y Axis from the Workspace submenu.**

   The x-y axis will appear on the workspace.

2. **Choose Numbers and Units from the View menu.**

3. **Switch the unit system to SI (radians)**

4. **Enter 1 into the Digits field.**

   The number of digits displayed to the right of the decimal will be set to 1.

5. **Click OK.**

6. **Choose Gravity from the World menu.**

7. **Click the None box.**

   Gravity will be turned off. This step eliminates both the need for the "floor" and the force calculations between the floor and the block.

8. **Click OK.**

9. **Choose Preferences from the Edit menu.**

   A dialog box with a series of check boxes will appear.

10. **Click the check box labeled "Automatic point equations on Object Snap".**

    Initially the box should be checked. After you click, the check should disappear, indicating that the equations feature has been turned off.

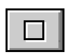
Creating the Components

This exercise has three mass objects, a 0.5 meter radius circle, a 2 meter long connecting rod and a 0.6 meter square block with a mass of 1 kilogram. The objects will be created, sized and initialized in the following steps. For clarity we will place the components along the x-axis. The block will be placed at the origin. All other positions will be referenced from the block.

Creating the Block

The block will be modeled by a square mass object. The square will be drawn with the Square tool. Its size, 0.6 meters, will be set with the Geometry utility window. It will be positioned at the origin using the Properties utility window.

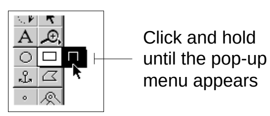
To draw the block:



1. Choose the Square tool.

Place the pointer over the Rectangle tool on the toolbar, click and hold until the Rectangle tool pop-up menu appears (see Figure 3-1 below). Choose the Square tool.

Figure 3-1
The Square pop-up menu



2. Drag out a square.

Position the pointer in the background, press and hold the mouse button down, move the mouse up and to the right. Now release the mouse button.

To size the block:

1. Select the square if it is not already selected.

2. Click in the Height or Width field in the Edit Toolbar and enter 0.6. Press Return.

Either field will change both the height and the width of the object so that it remains square.

To set the block's mass:

1. Select the square.

2. Choose Properties form the Window menu.

3. Click in the Mass field and enter 1.0.

To position the block:

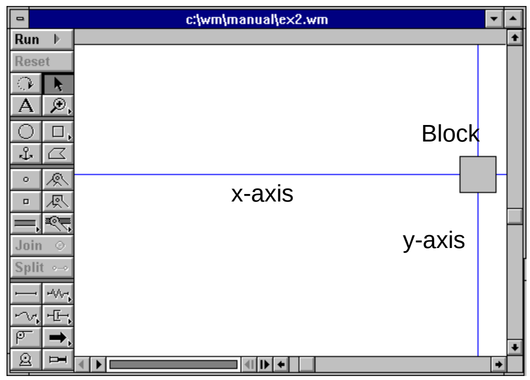
1. Select the square.

2. **Click in the X field on the Edit Toolbar and enter 0.**

3. **Click in the Y field and enter 0.**

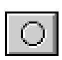
   The square is now at the origin. Use the scroll bars to position the block as shown in Figure 3-2.

Figure 3-2
The block at the origin



Creating the Driver Crankshaft

The crankshaft will be modeled by a circle. The circle will be drawn with the circle tool. Its radius, 0.5 meters, will be set using the Geometry utility window. Its position, 6.5 meters to left of the block, will be set with the Properties utility window. To draw the crankshaft:



1. **Click the Circle tool on the toolbar.**

2. **Drag out a circle.**

To size the crankshaft:

1. **Select the circle.**

2. **Click in the Radius field in the Edit Toolbar and enter 0.5.**

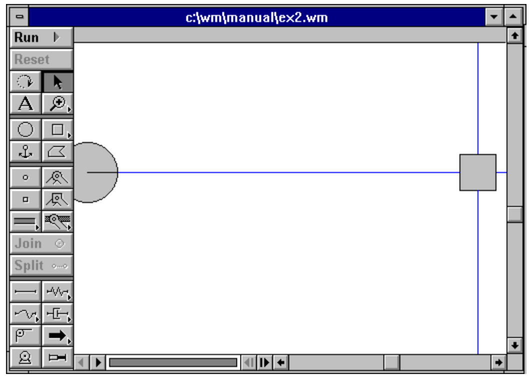
It is important to keep everything aligned vertically. To position the crankshaft:

1. **Select the circle.**

2. **Enter -6.5 and 0 into the x and y fields, respectively, on the Edit Toolbar.**

   Your screen should resemble Figure 3-3 below.

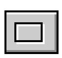
Figure 3-3
The crankshaft in position



Creating the Connecting Rod

The connecting rod will be modeled as a rectangle. It will be drawn using the rectangle tool. Though its height could be any value, it will be set at 0.35 meters to resemble the connecting rod in the problem. To set the rectangle size, the Geometry utility window will be used. Its position, 2.5 meters to the left of the block, will be set using the Properties utility window.

To draw the connecting rod:



1. **Choose the Rectangle tool from the Rectangle pop-up menu.**
2. **Drag out a thin rectangle.**

To size the rectangle:

3. **Enter 0.35 and 2 into the height and width fields, respectively.**

To position the rectangle:

1. **Select the rectangle.**
2. **Enter -4 and 0 in the x and y fields, respectively.**

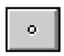
Creating the Elements for Joining

In this exercise, many points will be created. Most points will be created automatically when the tools for the spring, slot, and motor are used. Only the point and slot from the crankshaft-to-connecting rod joint will be created with element tools.

Creating a Point on the Crankshaft

The crankshaft needs to be attached to the connecting rod. The connection will be made using a point on the crankshaft and joining it to a slot on the connecting rod. The point on the crankshaft is positioned 0.2 meters from the crankshaft's center.

To draw the point on the crankshaft:



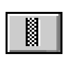
1. **Click the Point tool on the toolbar.**
2. **Click on the crankshaft.**

To position the point:

1. **Select the point if is not already selected.**
2. **Click in the X field and enter the value 0.2. Press the Return key.**
3. **Click in the Y field and enter the value 0.**

Creating a Slot on the Connecting Rod

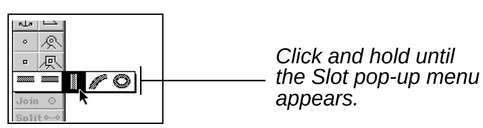
In the exercise the connecting rod has a slot on which the crankshaft slides. To model this, a slot will be placed on the connecting rod. To create the slot:



1. **Choose the Vertical Slot tool.**

   Choose the slot by placing the pointer over the Horizontal slot tool on the toolbar, clicking and holding until the Slot tool pop-up menu appears (see Figure 3-4 below). Choose the Vertical Slot tool to the right.

Figure 3-4
The Slot pop-up menu

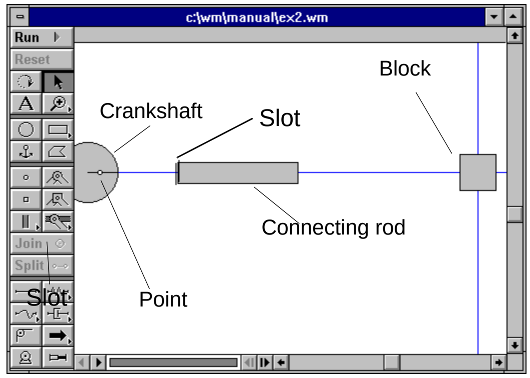


2. **Click on the Snap Point, indicated by the "X", on the center of the left side of the connecting rod.**

   A slot will appear.

   Your screen should look like Figure 3-5 below.

Figure 3-5
The connecting rod with the slot on it



Creating the Constraints

Exercise 2 requires four constraints: a keyed slot, a variable speed motor, a 50 N/m spring, and a slot on the connecting rod. The following steps will describe how to create these constraints with Working Model.

Creating the Keyed Slot

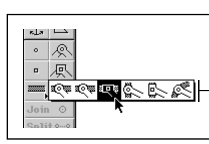
The connecting rod slides in a sleeve. The sleeve prevents the connecting rod from moving vertically or rotating. To model the sleeve, a keyed slot joint will be used. The Keyed Slot tool attaches a square point to the top mass object, and a slot to the object beneath the point (in this case it is the background), and then automatically joins them. The position of the keyed slot is not critical. It can be placed anywhere on the connecting rod.

To create and position the keyed slot:

1. **Choose the Horizontal Keyed Slot tool from the Slot pop-up menu on the toolbar (see Figure 3-6 below).**

   Place the mouse pointer over the slot tool. Click and hold the mouse button down until the pop-up menu appears. Choose the horizontal slot.

Figure 3-6
The Slot pop-up menu



Click and hold until the Slot pop-up menu appears.

2. **Click on the connecting rod.**

A slot will appear on the background. It will be joined to a point on the connecting rod (see Figure 3-7 below).

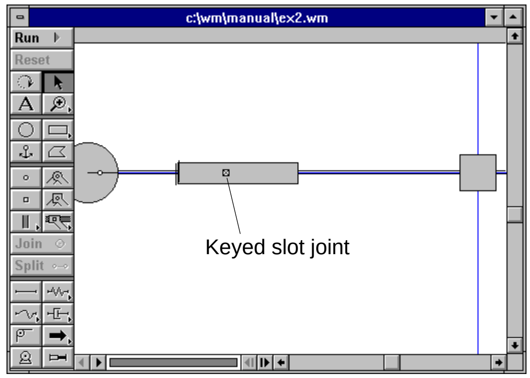


Figure 3-7
A keyed slot joint attaches the connecting rod to the background

Creating a Spring

To create the spring, the Spring tool will be used. The spring constant is preset at 50 N/m. To draw the spring:

1. **Click the Spring tool on the toolbar.**

2. **Click on the center of the right side of the connecting rod rectangle and drag to the center of the block. Release the mouse button on the block.**

   Use Object Snap to position the spring endpoints precisely.

   Your workspace should resemble Figure 3-8 below.

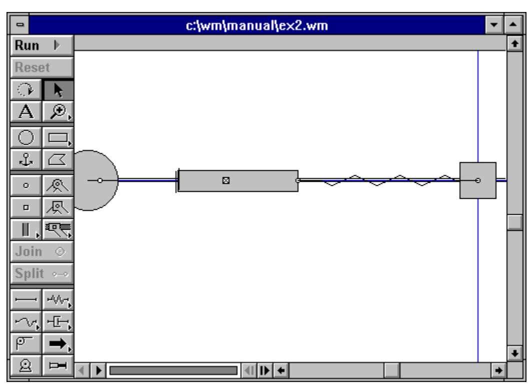


Figure 3-8
The spring attached is to the block and connecting rod

Creating the Drive Motor

The crankshaft is driven by a motor. Its position will be set using the Properties utility window

To draw the Motor:

1. **Click the Motor tool on the toolbar.**
2. **Click at the center of the circle when you see the “X”.**

   When a motor is created, two points are also created. One point is attached to the circle (the Point) and one is attached to the background (the Base Point).

Creating a Slider Input Device

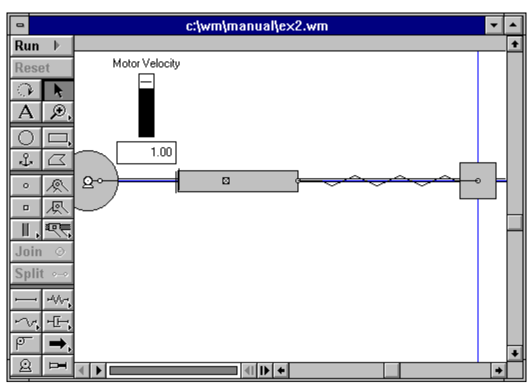
This tutorial asks for the rotational speed of the motor when resonance occurs. Different speed values could be entered via the Properties utility window, but a much more efficient way is to create an input device called a slider.

To create a slider:

1. **Select the crankshaft motor by box selecting it.**
2. **Choose New Control from the Define menu, and Rotational Velocity from the New Control submenu.**

   A slider similar to the one in Figure 3-9 below will appear on the screen.

Figure 3-9
A slider is created



Setting the Range of a Slider

The default range of a slider is not a proper one for the problem. To modify the slider range:

1. **Select the slider by clicking on its name.**
2. **Choose Properties from the Window menu.**
3. **Enter 10 in the Max field of the slider's Properties window.**

Joining the Connecting Rod to the Crankshaft

There are only two points in this exercise which need to be joined, the point on the crankshaft and the slot on the connecting rod. To create the joint between the connecting rod and the crankshaft:

1. **Select both the point on the crankshaft and the slot on the connecting rod.**

   The Join button in the toolbar will become highlighted.

   Use box select to select two objects. Alternatively, you can shift-select multiple objects. To shift select, select one object, and hold down the shift key to select another. Hold the shift key down to select more objects, if so desired.

2. **Click the Join button on the toolbar.**

Setting the Spring's Rest Length

When the connecting rod and crankshaft were joined, the connecting rod moved a substantial distance to the left. This movement stretched the spring, pre-loading it. To remove the pre-load, the spring's rest length will be reset to the larger current length.

To reset the springs rest length:

1. **Select the spring.**
2. **Choose Properties from the Window menu.**
3. **Click on the "(current)" value.**

   The value in the "length" field will automatically change to the current length. The spring's rest length will now be set to the current length.

Preventing a collision

The connecting rod and block will collide when resonance is achieved. The collision will dampen the motion of the block. To make the results of the simulation more interesting we will prevent the connecting rod-block collision.

To prevent a collision:

1. **Select the connecting rod and, while holding the Shift key, select the block.**
2. **Choose Do Not Collide from the Object menu.**

   The model is now complete and can be run.

3. **Click the Run button on the toolbar.**
4. **Move the slider bar up and down to see the effect on the rotation of the crankshaft.**

5. **Click the Reset button.**

Measuring Properties from the Simulation

It is interesting to watch the graphical motion of the driver and the driven objects as they interact in this simulation. To do so, a graph displaying two properties must be created.

Creating the Graph

The following steps describe how to create a graph of the block's position vs. time. The graph will be moved to the bottom of the screen. It will be enlarged so it reaches across the screen. To display the graph:

1. **Select the block.**
2. **Choose Position from the Measure menu, and X Graph from the Position submenu.**

   This meter will graph the x position of the block (driven mass) with respect to time.

To position the graph:

1. **Select the graph.**
2. **Click on the graph and drag it to the lower left corner of the screen.**

To size the graph:

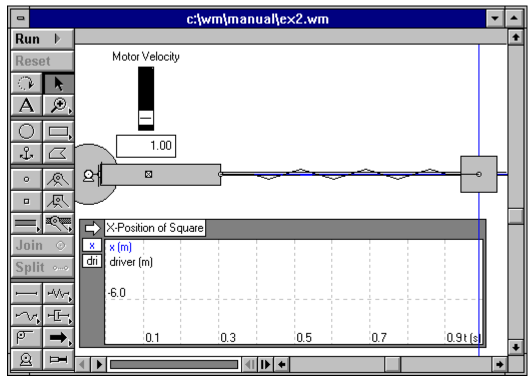
1. Select the graph.

Four squares (called handles) will appear at the corners of the graph indicating the meter is selected.

2. Click on the lower right handle and drag to the right side of the screen.

The graph will stretch to a larger size (see Figure 3-10).

Figure 3-10
An x-y graph is added



Adding a Parameter to the Graph

The second object of interest to graph is the connecting rod (driver). To add the parameter to the graph:

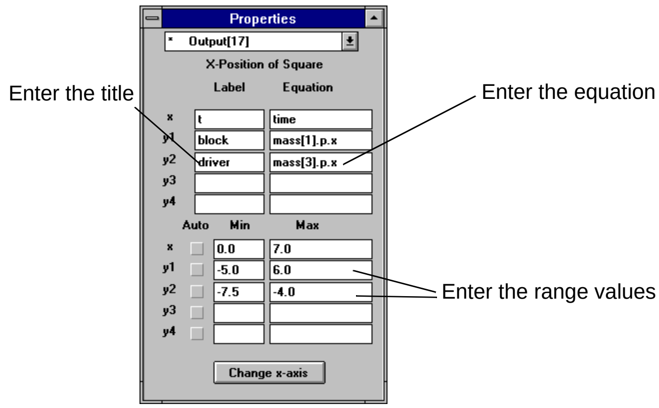
1. Select the graph.

2. Choose Properties from the Window menu.

3. Click in y2's Label field and type "driver" (see Figure 3-11 below).

This titles the new parameter. The text "driver" will appear in the y-axis of the graph.

Figure 3-11
Meter's Properties utility window



4. Press the Return key.

This enables the meter's y2 Equation field, allowing for editing of that field.

Note: some text must be entered into the Label field or the Equation field will remain disabled.

5. Click in the y2's Equation field and enter the following formula:

`mass[`***object id***`].p.x`

The ***object id*** *number for the connecting rod can be determined by placing the cursor over the connecting rod and reading the ribbon help at the top of the workspace.*

This will produce a graph of the values of the connecting rod's x-position.

Setting the Range of a Graph

Changing the x and y range of a graph can improve the quality of the graphical output. To set the range:

1. Select the graph.

2. Choose Properties from the Window menu.

3. Click on y1's Auto checkbox to turn off auto ranging.

4. **Enter various values into the Max and Min fields of y1 and y2 variables (see Figure 3-11 above). The following values work well:**

	Min	Max
y1	-5.0	6.0
y2	-7.5	-4.0

Experiment with different values and notice the changes in the graph. Be sure to turn off automatic range after you enter your variables or Working Model will modify the values.

Checking the Answers

Congratulations, the tutorial is now completed. Run the simulation, varying the motor speed until resonance is achieved.

The equation for the amplitude of driven harmonic motion is:

$$A = \frac{F}{\sqrt{m^2(\omega^2 - \omega_o^2)}}$$ where:

- A is the block's displacement amplitude
- F is the driving force
- m is the block's mass
- ω is the frequency of the driving force
- ω_o is the natural frequency of a spring-mass combination $\sqrt{\frac{k}{m}}$

As the frequency of the driver (ω) approaches the natural frequency of the spring/mass combination (ω_o), the amplitude approaches infinity. This condition ($\omega = \omega_o$) is known as resonance. For this exercise resonance will be achieved when the crankshaft's rotational velocity reaches $\omega_o = \sqrt{\frac{k}{m}}$, 7.1 rad/s.

Click in the slider's field and enter the value 7.1.

Now run the simulation.

4. Tutorial: Importing CAD Files

This tutorial will allow you to analyze an existing robot welder design by importing a CAD file into Working Model. The CAD file will be used for key geometries as well as accurate location of pin joints.

Tutorial 2 Concepts:

- Importing DXF files.
- Creating bodies in Working Model from lines that have been imported from a DXF file.
- Using the Join command to accurately build models.
- Creating and controlling actuators with slider input devices.

Introduction

Working Model's DXF import feature allows users to perform dynamic analysis on models drawn in CAD programs. With this feature, complex geometries and accurate pin placements from existing CAD files can easily be imported into Working Model. In addition, the ability to import DXF drawings allows users to take advantage of the sophisticated drawing tools found in high-end CAD packages

This example uses an existing DXF file exported from a popular CAD package. Virtually any CAD program that supports DXF files can be used, including AutoCAD®, Vellum®, CADKEY®, and MicroStation®. The tutorial walks the user through the import process and shows how bodies and joints are created. Slider controlled actuators are then added, and the robot-welder is put in motion.

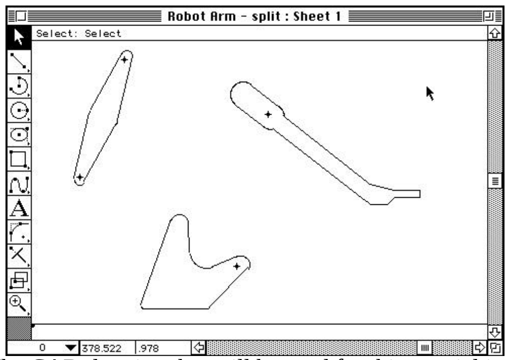
It is assumed that the user has already completed tutorial 1.

Preparing CAD Files for Export into Working Model

CAD drawings typically contain large amounts of detail that are not needed or used in a dynamics analysis. For example, rivets and screw threads may be included in a CAD drawing, but are not relevant to dynamic analysis and will slow down the simulation.

Before importing a CAD drawing into Working Model, users typically pre-edit to remove all but key body geometries and points where pin joints will be attached. In addition, large assemblies are split away from one another so that line segments can be simultaneously selected and converted into bodies once in Working Model. Bodies can also be created directly in the CAD package by creating closed polyline constructs before importing.

Figure 4-1
Robot Welder Drawing



The CAD drawing that will be used for this tutorial is shown in Figure 4-1. The drawing has been pre-edited to remove all but the key geometries and points. It is made up of arcs (including arc fillets), lines, and points. For this tutorial, we will not create closed poly-line constructs before importing.

Importing Into Working Model

The DXF file used in this tutorial is included on your demonstration disk, and is titled **ROBOT.DXF**.

To import the DXF file:

1. Open a new document.

Make sure all other documents have been closed.

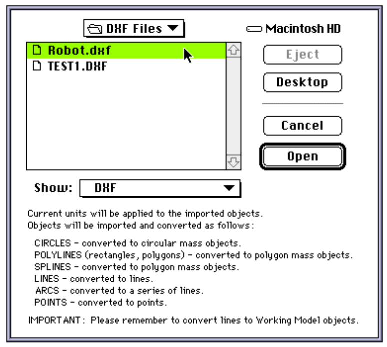
2. Select English (pounds) from the Numbers and Units dialog (selected from the View menu)

Before importing a file, it is very important to change to the unit system the CAD file was created in. While DXF files contain accurate numerical information on geometries, they carry no unit system information. If you forget to change to the correct unit system before importing, you may suddenly find yourself working with a 5 meter robot arm (vs. 5 feet)!

3. **Select Import from the File menu.**

   The import dialog box will pop-up as shown in Figure 4-2 (Macintosh version - Windows will be slightly different).

Figure 4-2
The Import Dialog box



4. **Locate and select the file named "ROBOT.DXF" from the dialog box, then click on the Open button.**

   The DXF file import will begin.

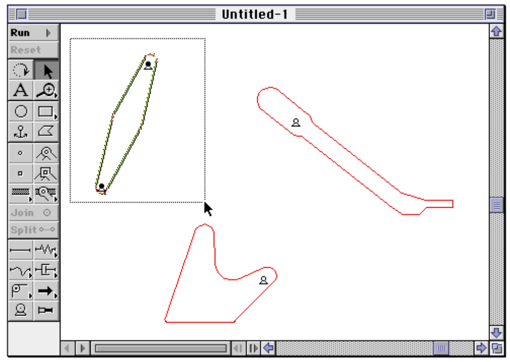
Working Model provides dialog boxes to inform the user as to the progress of the import. All DXF objects are first read from the file. Next, these objects are converted into Working Model entities. Finally, these new entities are merged into the existing Working Model document. Large DXF files may take as long as 1-2 minutes to read in and convert, depending on the speed of your machine.

Current unit settings (as given in the Numbers & Units dialog box) are applied for DXF files. For example, if an original CAD drawing had a 20-inch long object, and if you imported this DXF file under SI unit system (where length is measured in meters), then the object would become 20-meter long.

5. **Notice how objects have been converted.**

Working Model converts arcs and spline based DXF objects into a series of very small line segments. Although the objects you see in Figure 4-3 appear to have smooth, rounded edges and curves, they are actually made up of many individual lines segments. Points are attached to background (since DXF file does not have any information on which points belong to which objects) and show the accurate position of where pin joints will be located.

Figure 4-3
DXF File Imported to Working Model



Converting Imported Lines Into Bodies

This DXF file contained many small line segments and no closed poly-line constructs. Therefore, all groups of lines must now be converted into bodies which Working Model can use in a simulation.

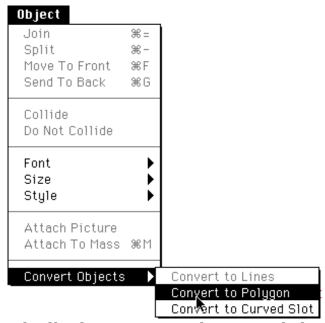
1. **Choose the arrow tool from the tool palette.**
2. **Box select the upper left hand grouping of lines (see Figure 4-3).**

   To box select, first click to create the first corner of the box, keeping the mouse button down. With the mouse button down, drag to create the box. Release when all line segments are enclosed in the box. The line segments will become highlighted as they are selected.

3. **Convert these lines into a polygon (which is a Working Model body) by selecting Convert Objects/Convert to Polygon from the Object menu.**

   See Figure 4-4.

Figure 4-4
Object Conversion Submenu



4. **Similarly, convert the remaining two groups of lines into polygons.**

Attaching Points to Bodies

Pin joints can be accurately created in Working Model by first specifying the exact pin locations with points in a CAD program. When points are imported into Working Model, they are initially attached to the background. They must then be attached to the appropriate body.

1. **Box-select the center body.**

   The box will enclose both the pins beneath the body, selecting the body and both pins. Multiple pins can be attached to a mass simultaneously.

2. **Select Attach To Body from the Object menu.**

   Select Attach to Body as shown in Figure 4-5. Attach To Body will attach all selected points to the selected body object without moving any of them.

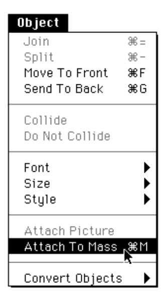


Figure 4-5
Attach to Mass menu item

3. **Similarly, attach the remaining pins to each body.**

Pinning Bodies Together with the Smart Editor

Designs can be accurately built by using Working Model's Smart Editor. In this tutorial, the Join feature of the Smart Editor will be used to create pin joints from the pins we just attached to each mass. The Smart Editor will automatically reposition a body during a Join command in order to make the pin joint. To prevent a body from moving or rotating during Join command, the anchor tool is used.

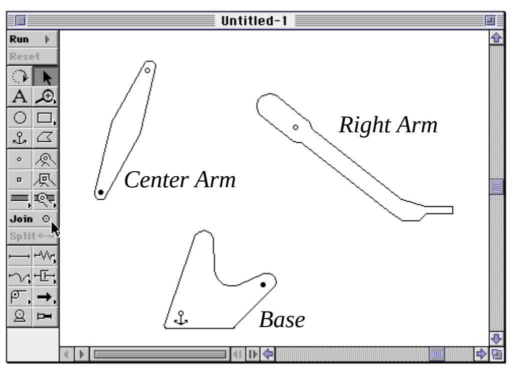
1. **Select the Anchor tool from the palette.**
2. **Place the Anchor on the base of the robot welder.**

   The base is the lower body. The anchor can be placed anywhere on the body. See Figure 4-6 as an example.

3. **Select the pin on the base by clicking the mouse on it. Holding the shift key down, select the left-most pin on the center arm of the robot welder.**

   See Figure 4-6. Holding the shift key down during selections allows you to select multiple objects. Note that the Join palette button is active.

Figure 4-6
Anchoring the base



4. **Create a pin joint with these two points by clicking on the Join command in the palette.**

   The center arm body will move in order to create the pin joint because the base has been anchored down. If neither had been anchored, the Smart Editor would move whatever is most convenient.

5. **Similarly, create a pin joint between the right most pin on the center arm body and the right arm body using the two remaining pins.**

   Both arms now move during the Join command because neither is anchored down. The base will not move or rotate.

Repositioning Bodies After Joining

Bodies have a rotational orientation that is defaulted to 0 (in the current unit system) when they are created. In order to easily attach actuators, we first must reorient the center and right arm to their default rotation.

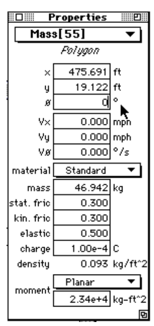
1. **Select the center arm body by clicking on it.**

   The center arm body will become highlighted.

2. **Change the rotational orientation of the center arm body to 0 in the ø field on the Edit Toolbar.**

   See Figure 4-7. Press tab after typing in a zero. The center arm body will move to the orientation it had when it was first created.

Figure 4-7
Properties Window

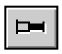


3. **Similarly, change the rotational orientation of the right arm body.**

   Smart Editor may decide to slightly rotate the center arm as well. It's okay. If this adjustment is annoying, you can temporarily anchor down the center arm before you set the right arm rotation to zero, and delete the anchor later.

Attach the Actuators to the Robot Welder

We will use two “length” actuators to control our robot welder. A “length” actuator takes a length as input, and provides the needed force at its endpoints to achieve this length. These actuators can be accurately placed with the Smart Editor if desired. We will position the actuators only roughly in this tutorial.



1. Select the actuator tool from the palette.

First, you will create the left most actuator as shown in Figure 4-8.

2. Click on the base. Hold the mouse button down and draw the actuator until it reaches the right arm. Release the button when it is in the correct position.

An endpoint is automatically attached to a body when the arrow is positioned over it during clicking of the mouse button.

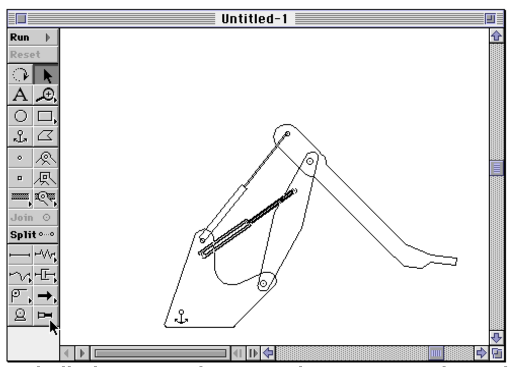


Figure 4-8
Robot arm with Actuators

3. Similarly, create the second actuator as shown in Figure 4-8.

Create Input Sliders to Control Each Actuator’s Length

We will now create input sliders to control each actuator’s length (these actuators can also be controlled with a formula, or even through a link to an external program like Excel or MATLAB).

To create the sliders:

1. Select an actuator by clicking on it.

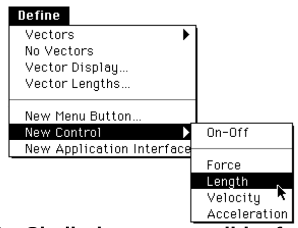
The actuator will become highlighted.

2. Choose New Control/Length from the Define menu.

See Figure 4-9.

A slider will appear on the screen.

Figure 4-9
New Control Submenu



3. Similarly, create a slider for the second actuator.

Change the Input Slider Ranges

The input slider ranges are defaulted to a range that is too narrow for this simulation. We will now change the range of each input slider. To do so, we will select both sliders and change their ranges in one shot.

1. Select the two sliders by shift-select or box-select.

If you choose to shift-select, click on the upper portion of each slider in order to select it. Four black squares (called selection handles) will appear when the slider is selected.

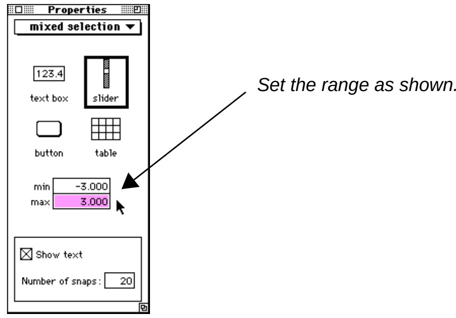
2. Open the Properties window from the Window menu.

The Properties window bears the name "mixed selection", indicating that more than one object are selected. The changes you make in the window will affect all the objects currently selected.

3. Enter -3 and 3 as the appropriate minimum and maximum range values. Use the Tab key to enter the value.

See Figure 4-10. These values will allow you to fully exercise your robot welder through a wide range of motion.

Figure 4-10
Properties Window for Actuators



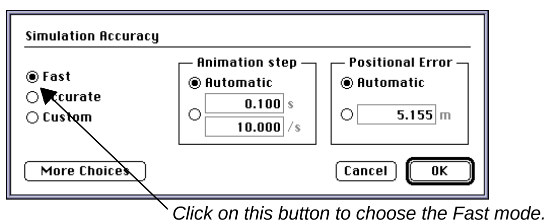
Running the Simulation

The simulation is now ready to run. In order to optimize the simulation speed:

1. Choose Accuracy in the World menu.

The Accuracy dialog box appears (see Figure 4-11).

Figure 4-11
Accuracy Dialog Box



2. Select Fast mode by clicking on the button in the Accuracy dialog box.

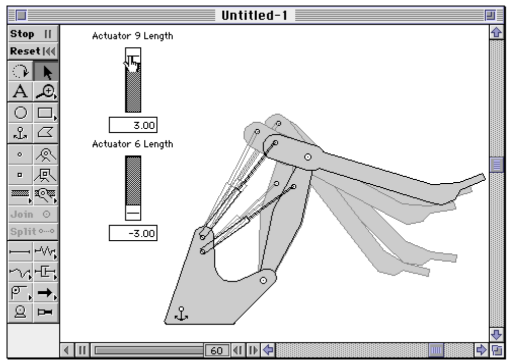
The fast mode will allow Working Model to present smoother and faster animation than the Accurate mode, although the latter provides higher accuracy.

1. Click on the Run button in the palette.

2. Move each slider through its range to see how it affects the robot welder’s motion.

See Figure 4-12. Note that we have changed the color of the robot welder using the Appearance dialog box available from under the Windows menu. In addition, we have turned on tracking from the World menu in order to take a “snapshot” at predetermined times during the simulation.

Figure 4-12
Running a simulation with the robot arm



3. **Using Meters, Measure the forces exerted by the actuators to reach the length you specify. Change the position of the actuator endpoints to see how the change affects the forces.**

   The force exerted by an actuator can be measured by first clicking on the actuator and then selecting the desired property in the ***Measure*** *menu. Pin placements can be repositioned directly with the mouse, or by typing in accurate values in the endpoint's Properties dialog box.*

Notes on Actuators

The length of actuators in the Properties window (or in the slider controls as shown in this tutorial) is specified as the offset from their initial length (i.e., the length when they were first created). Therefore, setting the slider to -3.0 does *not* make the actuator 3.0 meters long with switched endpoints. Instead, the actuator will be 3.0 meters shorter than when it was first created.

The physical length of an actuator cannot be negative. For example, if you drew an actuator of length 2.0 meters and set the slider bar for its length to be -3.0 meters, then the physical length of the actuator will end up being 2.0 - 3.0 = -1.0 meters. Such setting will cause the simulation to be spurious and inaccurate.

Notes on Inconsistent Constraints

You may notice that certain actuator lengths lead to impossible configuration of the robot arm. For example, in Figure 4-8, suppose the bottom actuator (connecting the base and the mid arm) retains the length as shown, while the top actuator is specified to be twice as long. Then the positions of the pin joints and actuator endpoints would be inconsistent.

In such cases, Working Model simulation will become spurious. You can have Working Model detect inconsistent constraints automatically by turning on the Inconsistent Constraint warning. To activate the warning:

1. **Choose Accuracy in the World menu.**

   The Accuracy dialog box appears.

2. **Click on the More button (on Windows) or More Choices button (on Macintosh).**

   The Accuracy dialog box expands to show more settings.

3. **Under warnings, turn on the Inconsistent Constraints.**

   A checkmark appears in the box to indicate that the inconsistent constraint detection feature is active.

More Analysis

You have now constructed a robot welder from an imported DXF format CAD file. The types of analysis you can conduct on this model are virtually unlimited. Check forces on pin joints, calculate the maximum force that a particular actuator configuration can lift, or even develop a controller using a program like MATLAB - then watch as you welder responds to this controller's commands.

The Next Step

Congratulations! You have now completed all of this guide's tutorials, and should have a good understanding of how powerful Working Model is.

The demonstration version of the software permits you to create your own simulations of mechanisms with up to five bodies. We encourage you to model several of your own mechanical systems to understand how Working Model can help you design more accurately and efficiently. If you have any questions during this process, a sales representative will be happy to assist you.

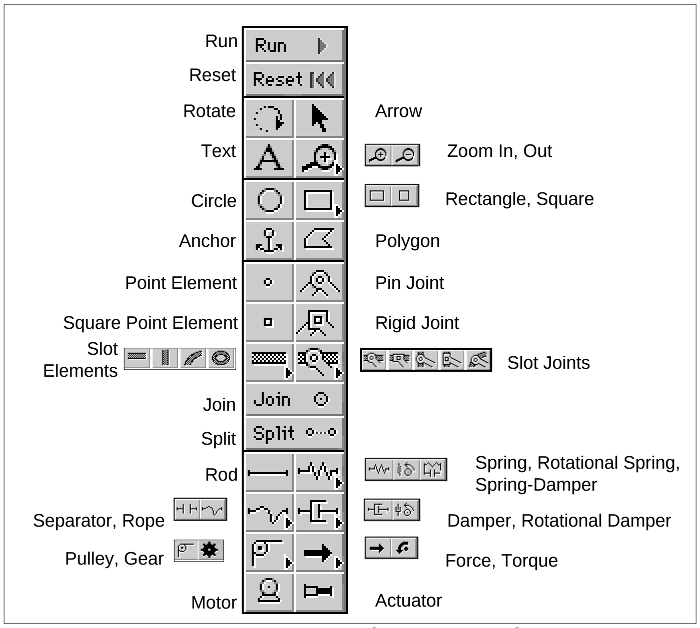
We hope you have enjoyed this demonstration copy. Please pass it on to a colleague or a friend when you are finished. When you are ready to place your order, please call us at the number on the back of this guide. We look forward to hearing from you.

5. Appendix

Toolbar

Every Working Model simulation window has a toolbar on the left side. The toolbar (Figure 5-1) provides access to the tools you will use to create simulations.

Figure 5-1
The Toolbar and Pop-up Palettes



The toolbar contains a series of boxes, each of which displays an icon that represents a specific tool. Some of these boxes are pop-up palettes (to save space and to provide easy access to a set of tools). Pop-up palettes are indicated by a small arrow in the lower right-hand corner of the box.

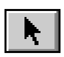
To open a pop-up palette, position the pointer on a box with an arrow in the lower right-hand corner, and press the mouse button. The pop-up palette appears to one side of the box. Drag the pointer to highlight the desired tool and release the mouse button. The last selected tool is displayed. You can always click on the icon to re-select the tool without opening the pop-up palette.

If you click once on a tool, it will be selected for the next operation; after that operation the selected tool will revert to the Arrow. If you want to use a tool for several successive operations, then you should double-click on it. This difference (single *vs.* double-clicking) will be indicated on the tool palette by color: a double-clicked item will be dark gray, while a single-clicked item will be light gray. To quickly select the Arrow tool, press the space bar. To quickly select the Rotate tool, press the "r" key.

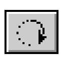
The Working Model toolbar consists of the following tools:

The **Run** button starts a simulation. When a simulation is in progress the Run button will be replaced with a Stop button. A click on the Stop button will stop the simulation.

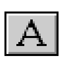
The **Reset** button is used to bring a simulation back to its initial conditions (the first frame).



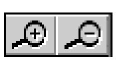
The **Arrow** tool is used to select an object or a group of objects, or to drag a selected group of objects on the screen. Pressing the space bar will automatically select the Arrow tool.



The **Rotate** tool is used to rotate an object or a selected group of objects. Objects can be rotated about their center of mass, as well as about pin joints and measurement points. You will see a line snap to the points about which objects can be rotated.   You can use a keyboard shortcut by pressing the "r" key to select the Rotate tool.



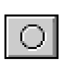
The **Text** tool is used to enter text directly onto the simulation workspace.



The **Zoom** pop-up palette has two selections: **Zoom In** and **Zoom Out**.

 The **Zoom In** tool increases the magnification of the workspace by a factor of two (2x). The new view is centered on the area around the pointer. Holding down the shift key will toggle this tool to the **Zoom Out** tool.

 The **Zoom Out** tool decreases the magnification of the workspace by a factor of two (1/2x). Holding down the shift key will toggle this tool to the **Zoom in** tool.



The **Circle** tool is used to create circular mass objects.

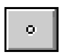
The **Rectangle/Square** pop-up palette has two selections: Square and Rectangle.

 The **Square** tool is used to create square mass objects.

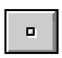

The **Rectangle** tool is used to create rectangular mass objects.


The **Anchor** tool locks the motion of mass objects. Anchored masses will not move unless an equation is entered to control their position. As a shortcut, you can press “a” to select the tool.

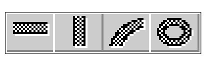

The **Polygon** tool is used to create irregularly shaped mass objects. Define each point with a single click. Double-click to signal the last point, or press the space bar after you single-clicked the last point. The polygon will automatically close, connecting the first point with the last point. You can later edit polygons graphically (choose **Reshape** in the **Edit** menu) or numerically (select polygon, and choose **Geometry** in the **Window** menu).


The **Point Element** tool is used to create a point element. Points can be used for measuring properties at a specific location on a mass object. A point can be combined with a slot or another point to form a joint (a slot joint or a pin joint). The **Join** command is used to combine Point elements into joints.


The **Pin Joint** tool is used to create a pin joint. A pin joint prevents separation between two mass objects at the location of the pin, but allows rotation. All joints can join a single mass object to the background.


The **Square Point Element** tool is used to create a square point element. A square point can be combined with a slot or another square point to form a joint (a keyed slot or a rigid joint.


The **Rigid Joint** tool is used to create a rigid joint. A rigid joint prevents movement or rotation between two mass objects.


The **Slot Element** pop-up palette has tools for creating horizontal and vertical slot elements.


The **Horizontal Slot** tool is used to create a horizontal slot element. A slot joined to a point forms a slot joint. A slot joined to a square point forms a keyed slot joint. (A description of keyed- *vs.* pinned-slot joints is found further below.)


The **Vertical Slot** tool is used to create a vertical slot element.


The **Curved Slot** tool is used to create a curved slot from a series of smoothly interpolated *(splined)* control points. Define each control point with a single click, and double-click the last point.

The **Closed Curve Slot** tools is used to create a slot consisting of a closed curve. Define each control point of the curve with a single click. Double-click to signal the last point. The curve will automatically close, connecting the first point with the last point.

The **Slot Joint** pop-up palette has tools for the horizontal, vertical, and curved versions of the Slot Joints**.** The straight slot joints may be pinned or keyed.

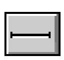
A **pinned slot joint** forces a point on one mass object to align with a slot on the second mass object or the background, while permitting rotation.

A **keyed slot joint** forces a point on one mass object to align with a slot on the second mass object or the background, while prohibiting rotation.

The **Join** button forms a joint from two elements. Select both elements—use the shift key to select multiple objects—and then click the **Join** button. The elements need not overlap. Joining two point elements will form a pin joint. Joining two square point elements will form a rigid joint. Joining a slot element with a point forms a pinned slot joint. Joining a slot element with a square point forms a keyed slot joint.

The **Join** button also recombines elements that have been separated using the **Split** button.

The **Split** button separates a joint into its component elements. Select a constraint and then click the **Split** button. A dashed line indicates the separated elements of a constraint. The **Join** button recombines constraint elements that have been separated using the **Split** button.



The **Rod** tool creates a massless, inflexible link between two mass objects. Rods cannot be compressed or extended. Rods can be attached between one mass object and the background, or between two mass objects. The endpoints of the Rod are its attachment points and function as pin joints.

The **Spring** pop-up palette provides three tools:

Springs resist stretching or compression. Springs can be attached between one mass object and the background, or between two mass objects (the endpoints of the spring are the attachment points and function as pin joints).

Rotational Springs produce a twisting force as they are wound up. Rotational springs can also be placed on top of a single mass object, in which case they will connect the mass object and the background. A rotational spring that is placed on two overlapping mass objects will be connected to both mass objects. Rotational springs have a built in pin joint.

Spring-Dampers provide a combination spring and damper. Spring-dampers can be attached between a mass object and the background or between two mass objects (the endpoints of a spring damper are the attachment points and function as pin joints).

The **Pulley / Gear** tool palette has two tools.

A **Pulley** tool creates a rope threaded through several pulleys. Define each pulley with a single click. Double-click to signal the last pulley. Any pulley within a link can be attached to either the background or to a mass object.

Since Working Model approximates pulleys as "thread holes" through which a rope is routed, they are massless and dimensionless.

A **Gear** tool connects any two mass objects with a gear constraint. Press and hold down the mouse button on one object, and drag the mouse pointer to the other to define a pair of gears.

By default, Working Model defines gear constraints to be external (spur) gears. You can define internal gears (one of the gears is inside the other) by choosing the option from the Properties window. The gear icon on one of the mass object changes to a gear showing internal teeth.

The **Force/Torque** pop-up palette has two tools:

A **Force** acts on the mass object to which it is attached. The point of application can be positioned anywhere on the mass object. The direction of the force can be fixed with respect to the background or to the mass object.

Torques apply a "twisting" force.

A **Motor** exerts a twisting force between two masses. Each motor has a built in pin joint. A motor can be placed on top of a single mass object, in which case it will connect the mass object and the background. A motor that is placed on two overlapping mass objects will be connected to both mass objects.

The **Actuator** tool creates an object that exerts a force between its endpoints. Actuators can be attached to two mass objects or to one mass object and the background. The endpoints of the actuator are the attachment points.

Using Formulas

This section briefly describes the Working Model formula language. Working Model allows you to enter formulas in most places you would typically enter a number. Formulas enable you to build custom forces and constraints and to dynamically control the behavior of objects. Formulas are also used to link input controls to the simulation, and to control the data displayed by meters and output devices. For an example of how a formula can be used in Working Model, please refer to the tutorial in Chapter 3, "Tutorial: Driven Oscillation".

Formulas in Working Model follow standard rules of mathematical syntax, and strongly resemble the equations used in spreadsheets and programming languages. Formulas are composed of identifiers, fields, operators, and functions. These are in turn described below.

Identifiers

Identifiers are used in formulas to identify an object. There are five basic types of identifiers.

```
body[3]         - {rectangles, polygons}

point[2]        - {point, square point}

constraint[44]  - {e.g. ropes, actuators}

output[12]      - {meters}

input[5]        - {controls}
```

The number in the bracket refers to the object ID. Each object in Working Model has a unique ID. To find an object's ID, double click the object to display the Properties window for that object.

Fields

Each identifier in the Working Model formula language can have fields, which are used to access the values of basic properties such as position and velocity. Fields are specified by listing the identifier, a period (.), and then the field name. For example, to access the moment of inertia of a body object with an ID of 3, you would enter the following:

```
body[3].moment
```

Sometimes you will use two fields in a row. To obtain the rotation of a body object, you enter the following formula:

```
body[2].p.r
```

The **body[n]** identifier has two-level fields. The first level, **body[2].p**, refers to the configuration (x-y position and orientation) of body object 2 and is a 3-component vector. The field **p** has the second level of field **r**, which represents the angle. As a result, **body[2].p.r** refers to the orientation of the **body[2]**. Shown below is a list of fields that can be used in the Working Model demonstration:

Type	Field	Returned
Vector	.x	number
	.y	number
	.r	number
Body	.p	vector
	.v	vector
	.a	vector
	.mass	number
	.moment	number
	.charge	number
	.staticfric	number
	.kineticfric	number
	.elasticity	number
	.cofm	point
	.width	number
	.height	number
	.radius	number
	.vertex[n].x	number
	.vertex[n].y	number
Point	.p	vector
	.v	vector
	.a	vector
	.offset	vector
	.body	body
	.force	vector
Constraint	.length	number
	.dp	vector
	.dv	vector
	.da	vector
	.p1	point
	.p2	point
	.force	vector
Output	.x	number
	.y1	number
	.y2	number
	.y3	number
	.y4	number
Input	(none)	number

Operators

Operators include all of the common algebraic symbols (+, -, >, =,<=,>=).

Functions

Functions take from zero to three arguments, and return a number or vector value. All functions accept their arguments in the form

```
function(arg1, arg2.....)
```

There are two kinds of functions available. Math functions perform standard mathematical operations. Simulation functions return information from Working Model simulations.

Math Functions

In the reference list below, the parameter types of the functions are shown as:

- a, b, c for numbers
- x, y, z for vectors.

Name and usage	**Output(return value)**
`abs(a)`	absolute value
`and(a,b)`	logical AND (1 or 0)
`angle(x)`	angle of vector
`acos(a)`	arc cosine
`asin(a)`	arc sine
`atan(a)`	arc tangent
`atan2(a,b)`	arc tangent of a/b
`ceil(a)`	ceiling
`cos(a)`	cosine
`exp(a)`	exponential (e^a)
`floor(a)`	floor
`if(a,b,c)`	if a then b else c
`ln(a)`	logarithm (base e)
`log(a)`	logarithm (base 10)
`mag(x)`	vector magnitude
`max(a,b)`	maximum of a, b
`min(a,b)`	minimum of a, b
`mod(a,b)`	remainder of a/b
`not(a)`	logical NOT (1 or 0)
`or(a,b)`	logical OR (1 or 0)
`pi()`	constant π
`pow(a,b)`	power (a^b)
`rand()`	random (between 0..1)
`sign(a)`	sign (1 or -1)
`sin(a)`	sine
`sqr(a)`	squared (a^2)

sqr(x)	vector 2-norm ($\|x\|^2$)
sqrt(a)	square root
tan(a)	tangent
vector(a,b)	creates 2-vector

Simulation Functions

Simulation functions are used to extract data from the simulation. These functions are used in the various meters and vectors of Working Model.

Name	Inputs	Output
constraintforce	number	vector
	number,number	vector
	number,number,number	vector
frame		number
frictionforce	number,number	vector
groupcofm	number	vector
kinetic		number
length	number,number	number
normalforce	number,number	vector
section	number,vector	number