Introduction

Welcome to Dynamix - the dynamical simulation add-on to Analytix.

In this introduction we cover:

- What is Dynamix?
- What you need to run Dynamix.
- How to use this manual.

What is Dynamix?

Dynamix is an add-in module to Analytix for performing dynamical simulation.

Without the Dynamix module, Analytix can only perform Inverse Dynamics. That is, if the motion is given, Analytix can compute reaction forces due to applied forces and the inertial pseudo-forces generated by accelerating masses.

With the Dynamix module, you can derive the motion due to known applied forces.

For example, you can sketch a pendulum, give it a mass, and suspend it in a vertical plane. You can then release it and have Dynamix compute its motion. Then you can animate the motion, graph positions, velocities, accelerations, or compute any of the reaction forces which Analytix is capable of.

Or you can sketch a suspension system, add springs and dampers and let it go from a perturbed position. Dynamix will then comute the motion of the suspension.

What you need to run Dynamix

To run Dynamix you need:

- An 80286 or 80386 based personal computer running the DOS operating system.
- A hard disk.
- A monochrome graphics monitor or color monitor.
- A Windows-compatible graphics adapter card.
- A Windows-compatible mouse or pointing-device.
- Analytix version 2.0 or higher.

About This Manual

This manual is divided into two sections: an **Examples** section and a **Reference** section. The Analytix manual should also be referred to.

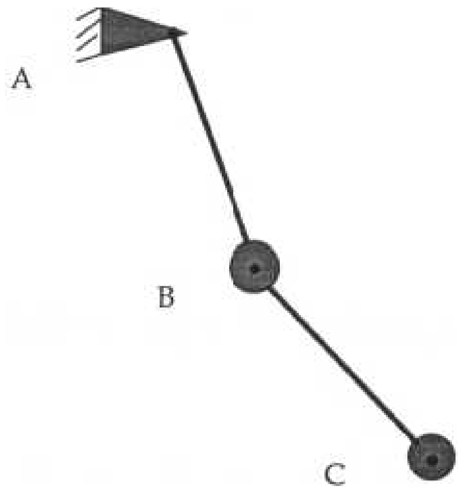
The Examples section consists of a sequence of examples of dynamic simulation with Analytix/Dynamix. The examples start simple and become gradually more complex. Hence the example section may be used as a tutorial.

The reference section documents the additional features which Dynamix adds to Analytix. These comprise in total three additional menu options.

Dynamix

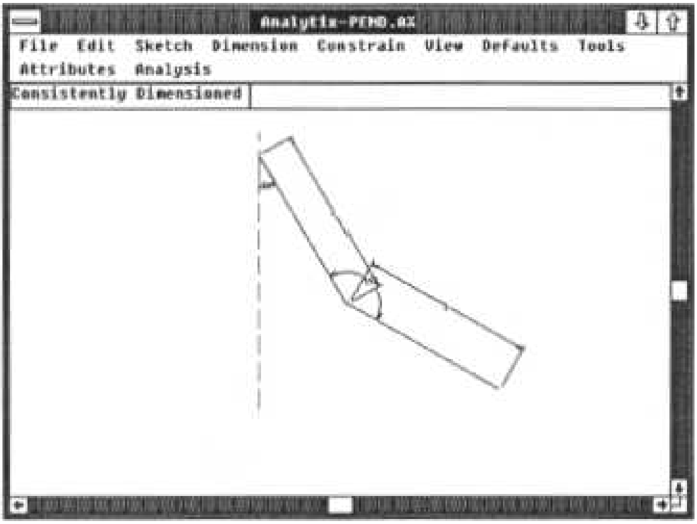
Examples

Example 1: A Double Pendulum



The Problem:

Our first dynamic simulation example is to simulate the double pendulum shown above which is hinged at A and B. Each arm of the pendulum is 1 m long and B and C each has mass 1 kg. We assume that the bars joining the masses are rigid and of negligible mass. The pendulum is initially released from rest with AB at angle 30 degrees to the vertical and angle ABC 150 degrees.



The Model

We sketch the pendulum in its initial position. The lengths of the bars are dimensioned as are the two angles specified in the problem.

Three further steps remain before dynamic simulation may be performed:

1 - The vertical construction line and the point where the upper bar of the pendulum meets this line must be specified as fixed.

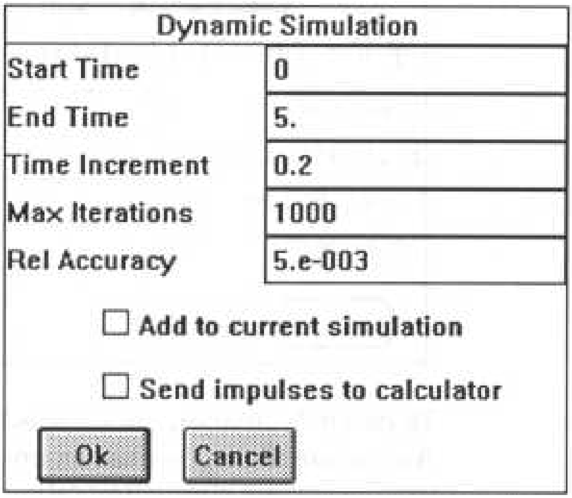
2 - The units should be set to SI and the drawing set to being in the vertical plane - this is done from the Defaults/Dynamics Defaults menu option.

3 - Masses must be given to points B and C. This may be done by double clicking on the point when in Select Mode. This brings up the Info box for that point, which is where the mass is entered.

The Solution

We have specified the initial geometry of the double pendulum by the lengths of the bars and by two angles. Before Analytix can do a dynamic simulation, it needs to know which dimensions are to be free to move in response to dynamic forces. In this example, the two angles will be free, but the lengths of the bars will stay fixed.

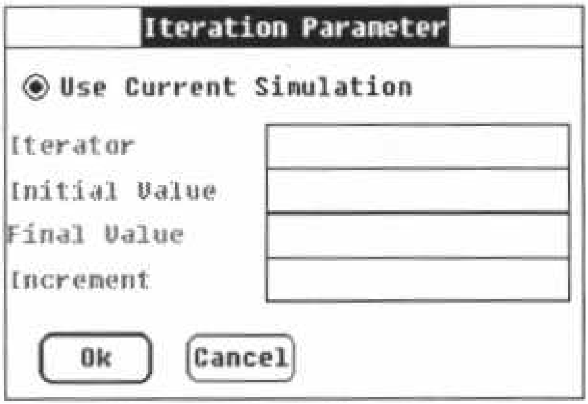
To perform a dynamic simulation, select the angle at A, hold down the Shift key and select the angle at B. Both these angles should now be selected (and highlighted). Now choose Dynamic Simulation from the Analysis menu.



The Dynamic Simulation Dialog Box lets you set the start and end times of the simulation and lets you specify at what time interval you wish to record snapshots of the motion. (Note that this interval is not the integration step size - the integration step size is set automatically by the system, but is always smaller than the snapshot interval.) You can also set the maximum total number of integration steps and the desired relative accuracy. (For more information see the reference section on Analysis / Dynamic Simulation

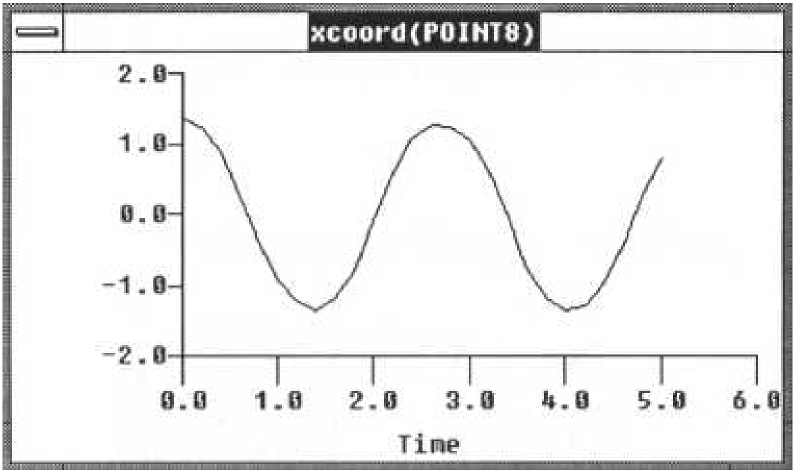
For the pendulum, simulate from time 0 to 5 with snapshots every 0.2 seconds.

As the simulation progresses, the drawing updates, but at somewhat less than interactive speed. However, at 0.2 second intervals the state of the system is stored into the current simulation. When the dynamic simulation process is complete, you can go back and make an animation of the simulation, or an envelope, a table or a graph.

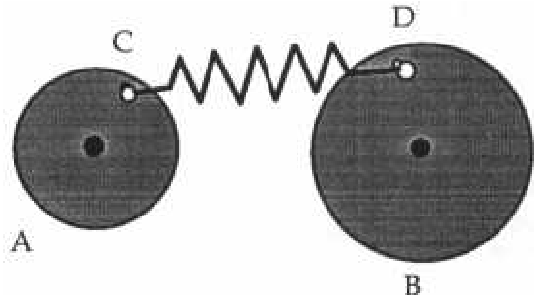


To do an Animation, pick Tools/Animate. The Animation dialog box has an option which lets you use the current simulation rather than specifying explicitly the dimension or variable to animate.

A similar option in the Table dialog box enables us to tabulate then graph the x coordinate of point B. .


xcoord(POINT8)
2.0
1.0
0.0
-1.0
-2.0
0.0
1.0
2.0
3.0
4.0
5.0
6.0
Time

Example 2: Springs and Drivers



The problem

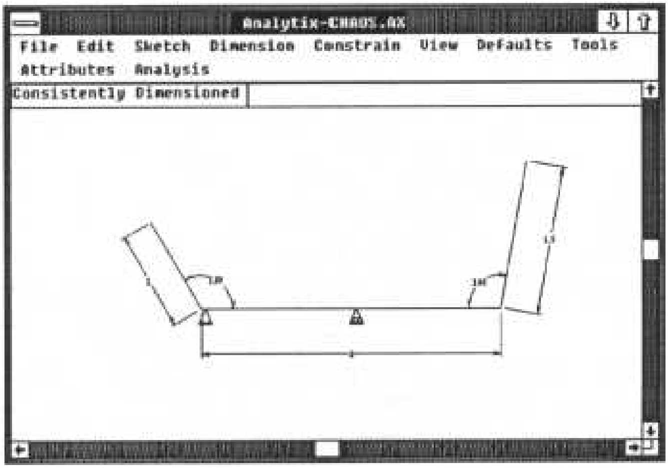
In the above example, flywheel A is driven at an angular velocity of 2 rad/sec. Flywheel B is free to rotate and is connected to A by a spring. We are to model the motion of the above system and graph the angular velocity of A during the first 10 seconds of the motion.

Flywheel A has radius 1 meter, B has radius 1.5m and the centers of the wheels are distance 3 meters apart. Flywheel B has moment of inertia 4kgm^2. The spring has unextended length 3.5m and stiffness 4N/m.

B is initially at rest, and the angle CAB is initially 120 degrees while DBA is 100 degrees.

The model

This example may be modelled by three lines: the horizontal line joining the centers of the two wheels and the lines joining each center to the spring attachment point of that flywheel.



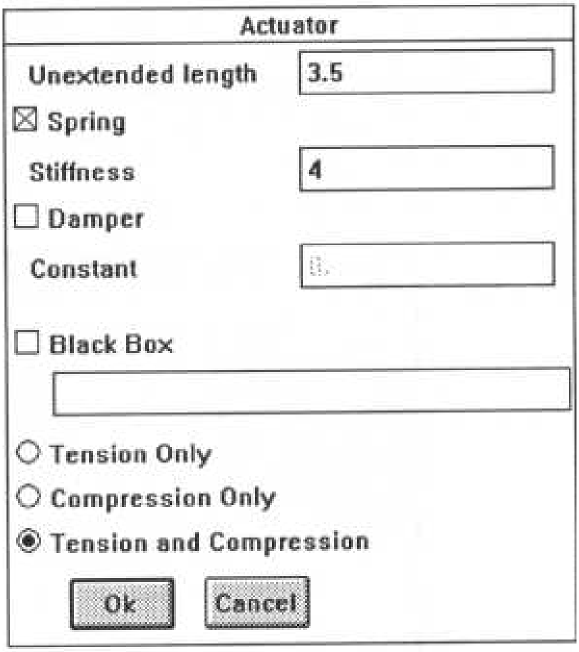
The horizontal line should be fixed as should the point A.

A moment of inertia should now be given to the line BD (this represents the inertia of the flywheel B.) Double clicking on the line brings up the information box, where the inertia can be added.

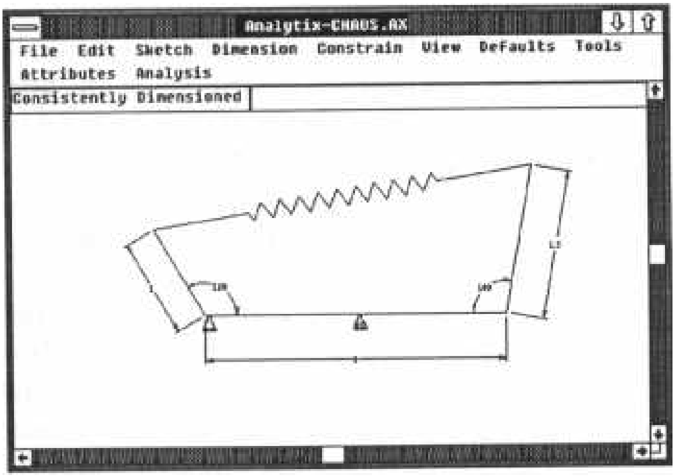
The angle at A should be given a velocity of 2.

To complete the model, we need to add a spring. Select the two points which the spring is to be attached to (select the first point, then hold down the Shift key and select the second point.) Now choose Add Actuator from the Analysis menu. You will be presented with the Actuator dialog box.

Actuators may be any combination of spring, damper and actuator. *For* this model, we wish only



a spring. Ensure that the Spring box is checked, and that the Damper and Black Box buttons are not. We



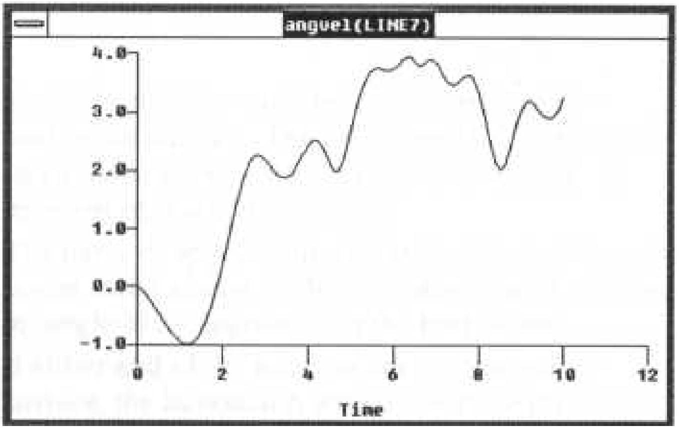
now need to enter its unextended length and stiffness.

Some springs work only while under tension; it is also useful to be able to have springs which work only when under compression. We assume our spring acts both while under tension and while under compression. Hence we make sure the Tension and Compression button is selected.

The solution

The only dimension which is free to move in response to dynamic forces is the angle at B. Although angle A moves, its motion is prescribed. Hence when we start up the simulation, we select only the angle at B, then choose Analysis/Dynamic Simulation.

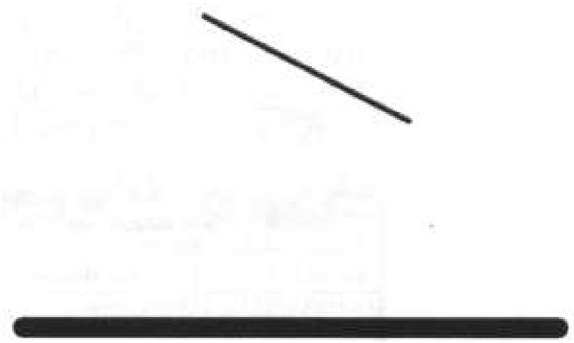
(if a dimension which has not been selected as free to move does have a non-zero velocity or acceleration, Analytix will assume that dimension is a driving



element and model it as moving with the prescribed acceleration and initial velocity, regardless of the dynamics of the model. A selected dimension on the other hand will have its acceleration dictated by the model's dynamics.)

Above is depicted a graph of the angular velocity of flywheel B over a 10 second interval.

Example 3: A Pin Dropping



The problem

In this example we model a rod bouncing on a horizontal surface. The rod is length 1 meter and has a mass of lkg with centroid at its center. Its moment of inertia is 0.2 kgm^2.

The bar is dropped from rest such that its lower end is one meter above the horizontal surface and it is at an angle of 60 degrees with the horizontal.

If either end of the rod touches the horizontal surface, the interaction with the surface is to be modelled as an elastic-plastic collision with coefficient of restitution 0.8.

The model

The geometric model consists of a horizontal and vertical construction line from which to reference the location of the falling bar, and a point positioned at the intersection of the construction lines to serve as the fixed point. The bar itself is represented by a line with a point located at its center of mass.

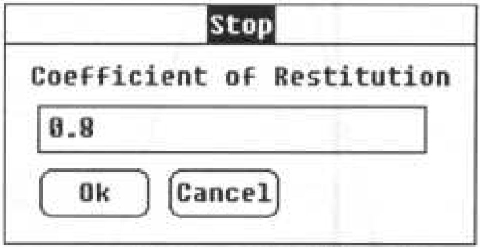
Distance Line/Point dimensions should be used between one end of the bar and the horizontal and vertical construction lines. The angle between the bar and the horizontal should be given. The rest of

the figure should be dimensioned as shown.

The horizontal line and the point at the intersection of the construction lines should be fixed. The line representing the bar should be given a moment of inertia and the point at its center should be given a mass. The Dynamics Defaults menu option should be used to set the drawing to be in a vertical plane.

To complete the model, we need to specify the fact that the endpoints of the bar are to bounce off of the horizontal surface. To do this we specify "stops" between these points and the horizontal construction line.

First select one of the endpoints, then hold down the Shift key and select the horizontal line. Now select



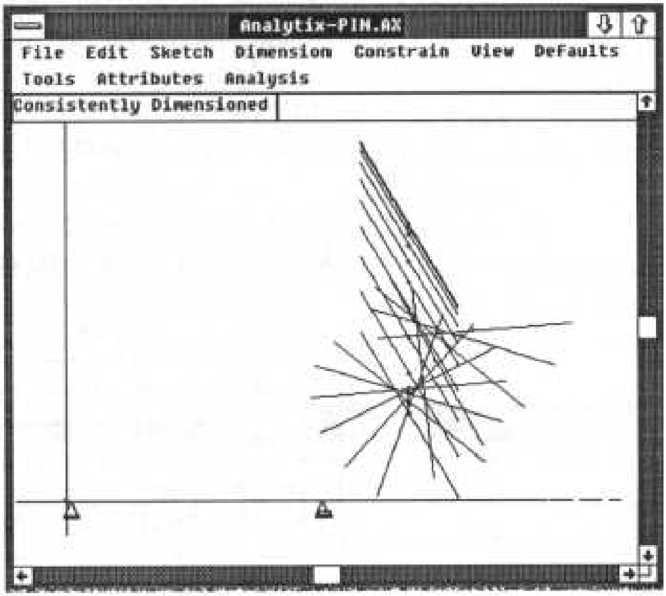
the Analysis / Add Stop menu option. The Stop dialog box now lets you enter a coefficient of restitution for any collision between the point and the line.

A coefficient of restitution of 0.8 implies that the impulsive forces during the collision will be responsible for generating a relative velocity after the collision of 0.8 times the relative velocity before the collision and in the opposite direction.

To complete the model, we create a second stop between the point at the other end of the bar and the horizontal line.

The solution

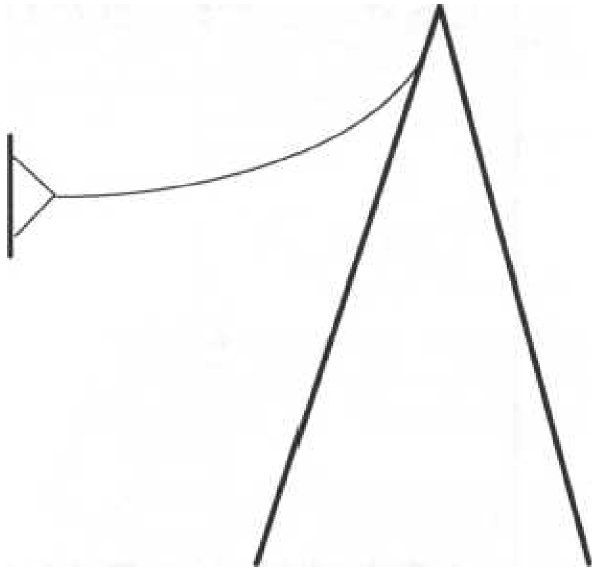
We create a dynamic model of the first second of the motion of the bar recording every 0.05 seconds. We select three dimensions as free to move in response to dynamic forces: the distance of the endpoint of the bar from the horizontal and vertical axes, and its



angle.

In the screen shot above, we see an envelope of the motion.

Example 4: A Child's Swing



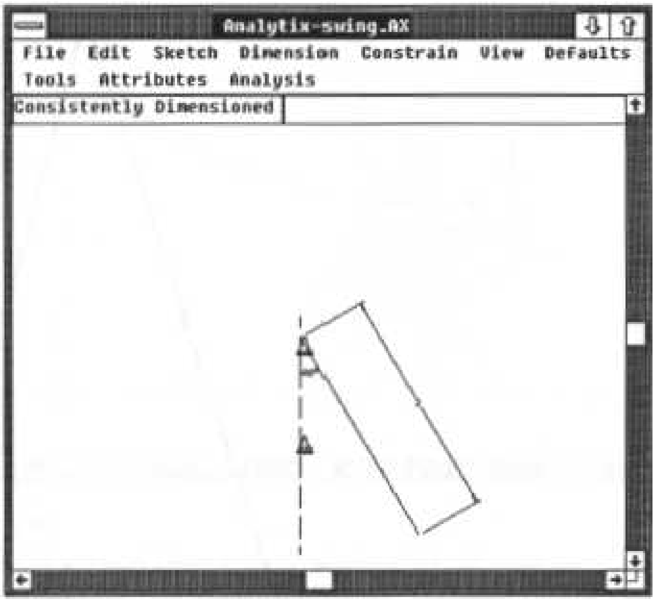
The problem

In this example, we model the pendulum action of a child's swing, taking into account the fact that the rope which supports the swing is incapable of withstanding compression.

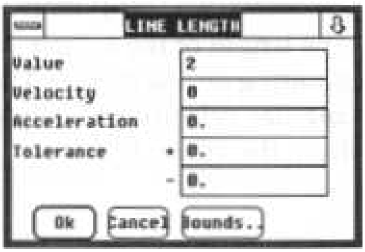
The length of the rope is 2 meters and a 30 kg child is sitting on the swing. The swing is given an initial angular velocity of 3.5 m/s at a position 30 degrees from the vertical.

The model

The geometric model is a vertical construction line and a line representing the rope supporting the

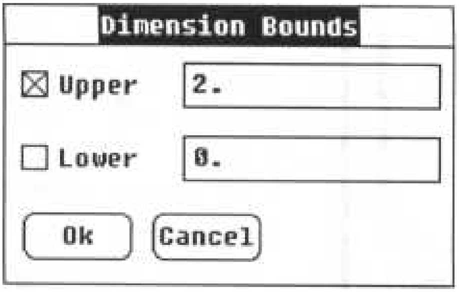


swing. The point at the end of the rope is given a mass of 30. The angle between the rope and the vertical is given a velocity of -3.5.



The rope in this model acts as a constraint on the motion so long as it is under tension. When it ceases to be under tension, it ceases to be a constraint.

We specify this behavior by specifying bounds on the variation of the length of the rope. It has a



maximum length of 2 meters. We do this by clicking on the Bounds button of the Dimension Info Box for this length. The Dimension Bounds Box now appears.

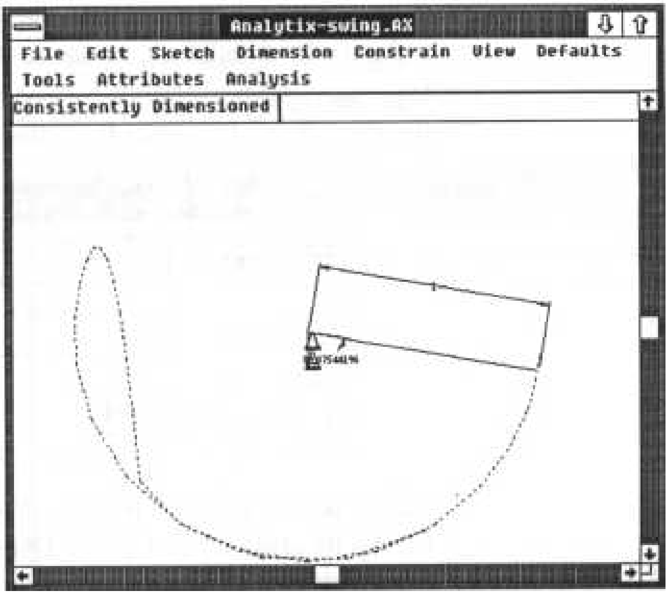
We check the Upper bound box and enter a value of 2. There is no lower bound for the length of the rope, hence we do not check the Lower bound box.

The solution

The dimensions which may change in the simulation are the angle which the rope makes with the vertical and the length of the rope (although this will stay constant so long as the rope is under tension.)

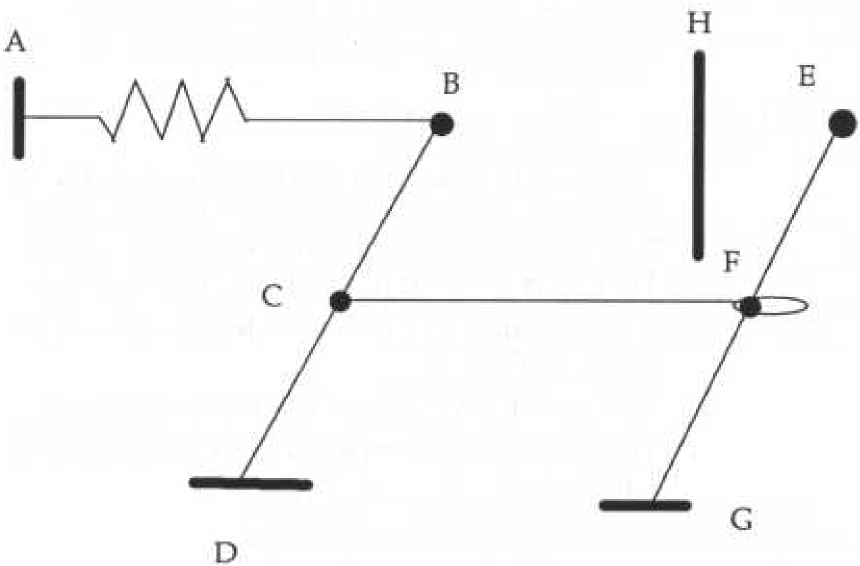
We run the model for 3 seconds saving every 0.1 second.

We note from the trace of the end point that the rope stays taught until high on the swing, when it collapses inwards and the swing falls freely for a short time. When the rope's length becomes 2, it pulls taught. The impact is modelled as a perfectly plastic interaction.



After selecting the point representing the swing's seat, Tools/Trace with the Use Current Simulation option gives a displacement curve for the swing.

Example 5: A Spring Actuated Mechanism



The problem

The spring actuated mechanism depicted above serves to establish contact between point E and the line at H. Bar CF has a half inch slot at F which allows bar BD to rotate a small angle before bar EG is engaged.

The collision between E and line H is to be modelled as an elastic/plastic collision with coefficient of restitution 0.25.

A	B	C	D	E	F	G	H
(-6,4)	(4,4)	(2,2)	(0,0)	(10,4)	(8 ,2)	(6,0)	(6.5,*)

The coordinates of the points in the diagram are shown above.

Spring AB has free length 5 inches and stiffness 20 lb/in. Bar BD has weight 0.2 lb centered at C and moment of inertia 0.3 lb.in^2. Bar CF has weight 0.31b and inertia 0.2 lb.in^2. Bar EG has weight 6 lb centered at F and moment of inertia 8 lb.in^2.

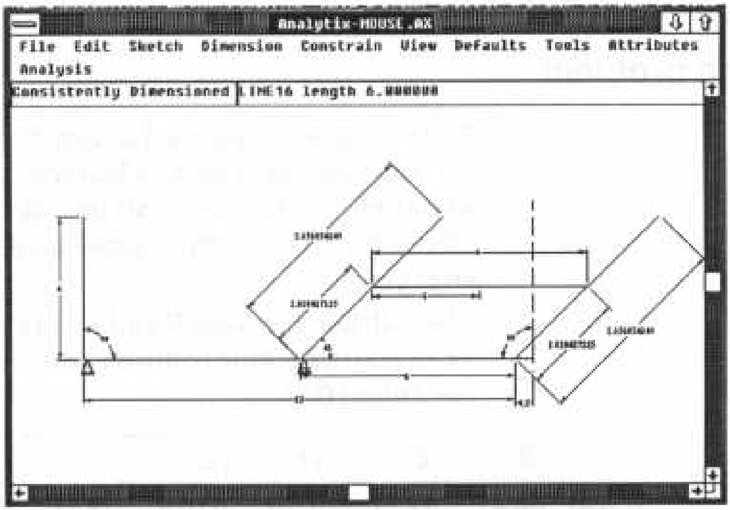
The model

The model contains vertical line from A to the level of D and G and a horizontal line through D and G. There is also a construction line representing the line H. This is the fixed framework for the mechanism.

The mechanism itself is modelled by the lines DB, GE and CF.

Dimensions are added to this model as shown.

Five further steps prepare the model for dynamic simulation:



1 - Use Defaults/Dynamics Defaults to ensure the system is set up to use ips units and a vertical orientation.

2 - Enter weights and moments of inertia.

3 - Create the spring between points A and B. (Select the two points then use Analysis / Add Actuator).

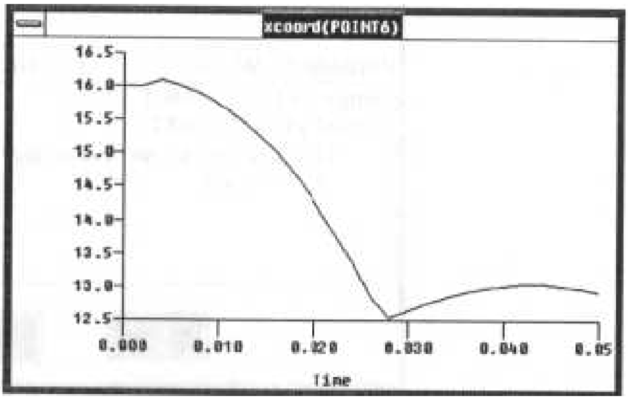
4 - Create a stop between point E and line H. (Select the point and the line and use Analysis / Add Stop).

5 - Set the bounds for the length of line CF (this can vary between 6 and 6.5 inches). (Select the dimension and use Attributes / Info to bring up the info box. Now click on Bounds.)

The solution

The two dimensions which may vary in the simulation are the angle at D and the length of CF (within the prescribed limits). Run the simulation for 0.05 seconds recording every 0.002 seconds.

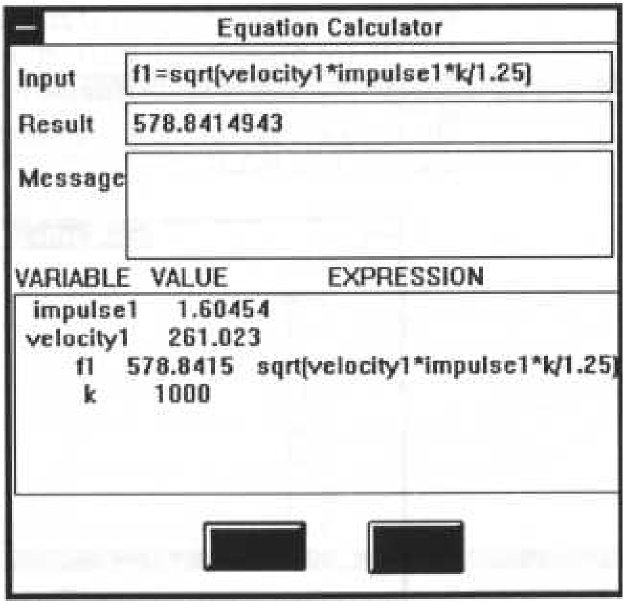
Shown is a graph of the x coordinate of point E during the simulation.



Impact Force

If you select the Send Impulses to Calculator option in the Dynamic Simulation dialog box, then both the impulse and the closing velocity for any collisions will be sent to the calculator. The impulse at the first impact has the variable name impulsel, the velocity has the name velocityl. Subsequent impulses have names impulse2, impulse3, etc. Subsequent velocities are called velocity2, velocity3 etc.

The impulse is a measure of force times time over the (short) duration of the impact. To calculate the maximum force we need some extra assumptions about the nature of the impact to allow us to compute its duration. One type of assumption would be to model the impact as the action of a spring with stiffness k. If I is the impulse, v the closing velocity and c the coefficient of restitution,



then the maximum force F_{max} may be derived from the following formula:

$$F_{max} = sqrt(v.I.k/(1+c)$$

Joint Friction

As a further refinement of the model, you may wish to simulate friction at the joints of the mechanism. You can do this using a rotational actuator.

To create a rotational actuator, select two lines then pick Analysis/Add Actuator.

An initial linear model of friction can be obtained by using a rotational damper. A more complicated friction model may be obtained by using a Black Box actuator.

Example 6: Planetary Dynamics

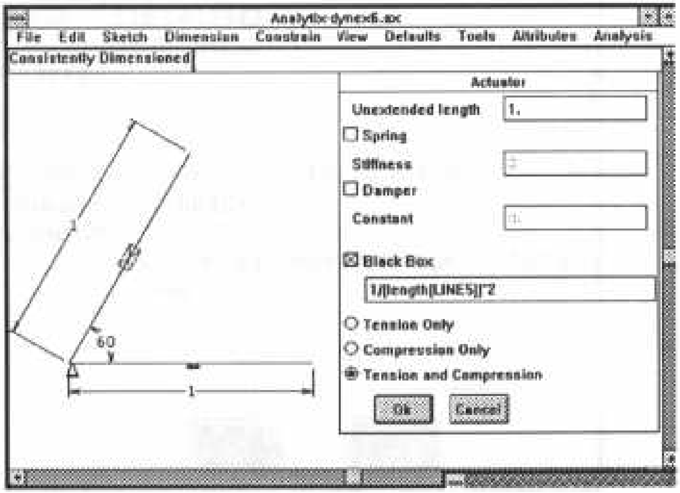
The Problem

As an example of the use which may be used of "Black Box" actuators, we make a simple model of Newtonian dynamics in the large.

The Model

We create a fixed point (the sun), a fixed reference line and a second line joining the fixed point to our planet. We create an actuator between the sun and the planet whose force is proportional to the inverse square of the distance between them.

In this model, we will look only for a qualitative picture of planetary dynamics, and will scale our units such that the gravitational constant times the mass of the sun is 1, the initial distance of the planet from the sun is 1, and the mass of the planet is 1.



We start the simulation with the planet moving at right angles to the line joining it to the sun, at such a speed that the line is rotating at 0.75 rad/time period. Hence the angle is given a velocity of 0.75.

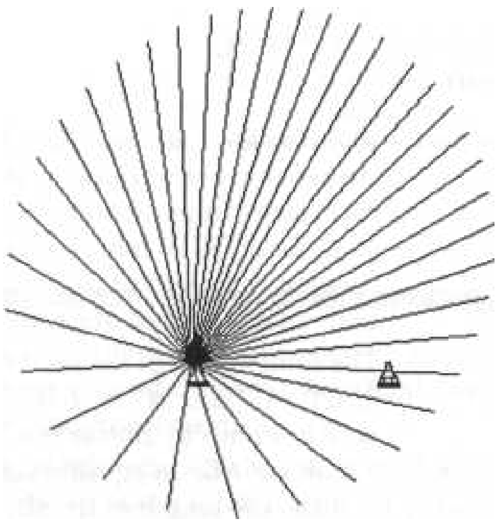
The Solution

To perform the dynamic simulation, we first ensure that the drawing is horizontal using the Dynamics Defaults menu option. (Gravity is modelled explicitly here by the inverse square actuator).

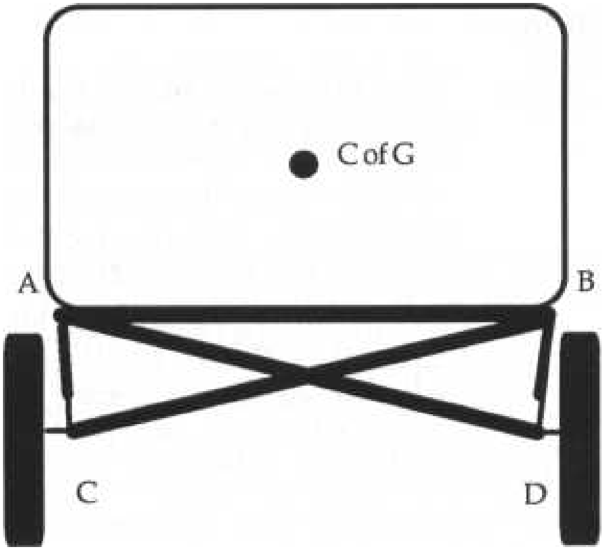
Both the angle dimension and the distance between the planet and sun should be free to move.

We perform a dynamic simulation for a time period of 3.5, saving at intervals of 0.1.

Below we see the characteristic elliptical orbit with the sun at the focus, and speed increasing the closer the planet is to the sun.



Example 7: A Suspension System



The Problem

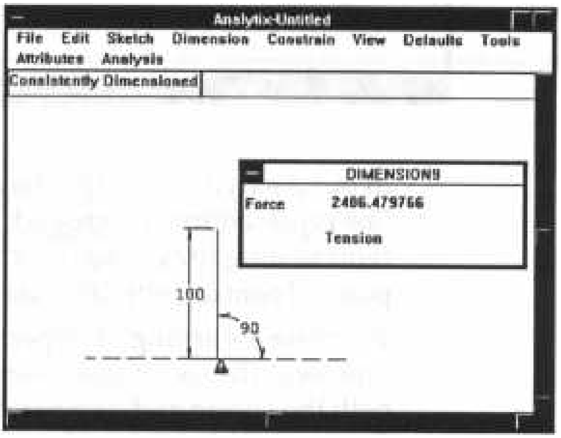
We wish to model the dynamic behavior of the suspension system shown above at the onset of cornering. In particular we look at its behavior at 40ft/sec on a 100 foot radius curve, where the suspension is initially at equilibrium and the cornering is started abruptly.

The dimensions of the model are as follows. AB is 5.75 ft, AD and BC are 6 ft. The weight of the vehicle which is born by this axle is 50001b and the C of G is 3 ft above AB. In equilibrium with the weight of the vehicle, the length of the shock absorbers at AC and BD is 1.5 ft. The shock absorbers have stiffness 5000 lb/ft, natural length 2 feet and damping coefficient 5000 lb.s/ft.

The Model

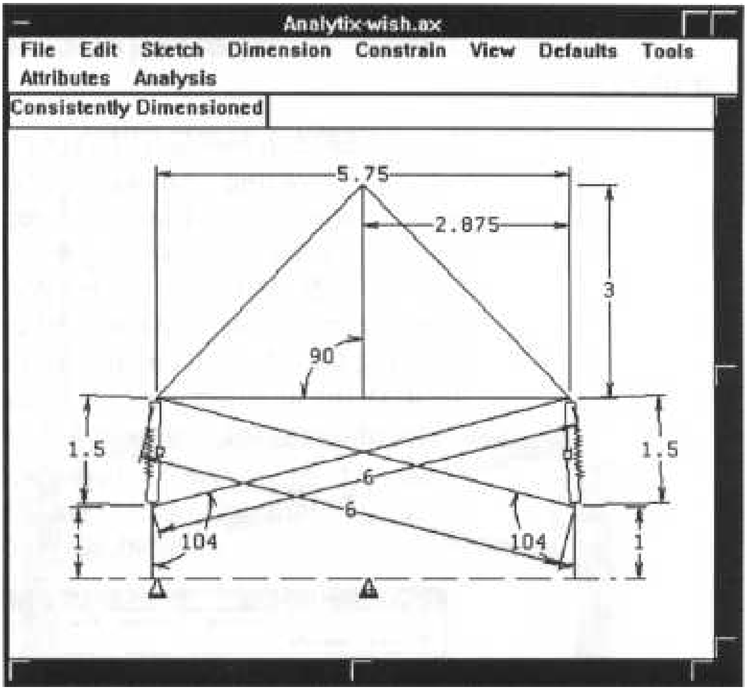
We first need to compute the side load due to the vehicle's cornering. An easy way of doing this is by using Analytix. We draw a horizontal construction line and a vertical line joining the center of the turning circle with the vehicle. We give the point representing the vehicle a weight of 50001b (first ensuring that the units being used are fps and that the drawing alignment is horizontal). We fix the center of the turning circle and the construction line.

We need to give the angle between the two lines in the drawing the appropriate velocity such that the velocity of the vehicle is 40, that is 0.4 rad/sec.



The required force is obtained by selecting the length dimension and choosing Resultant Force/Torque from the Analysis menu.

This is the force which will be applied to the center of gravity in our model of the suspension system.



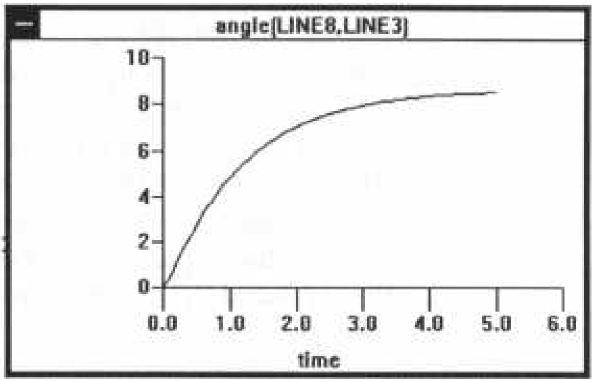
In our Analytix model we have added a construction line representing the ground, and two lines representing tires. The construction line and its point of contact with the outer tire are fixed.

To create the spring/damper combinations which represent the shock absorbers, we make sure that both the spring and damper selections are made in the actuator dialog box.

We run a dynamic simulation of the first 5 seconds after the onset of the cornering force.

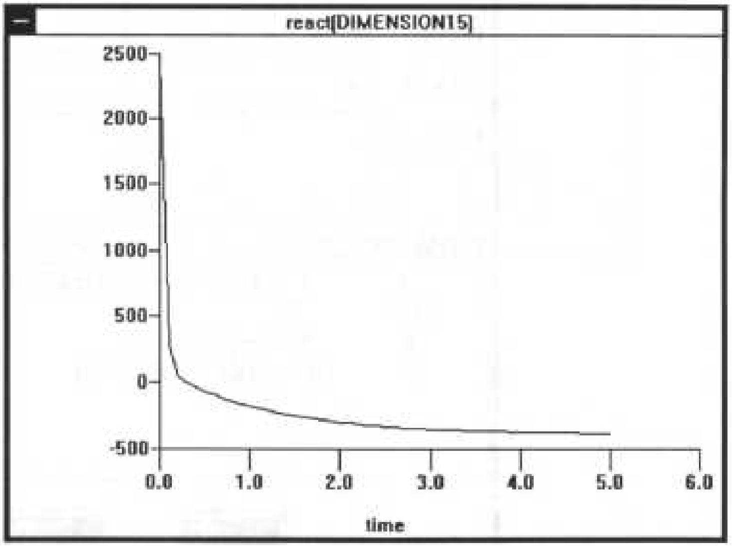
Our first output graph shows the roll angle as a function of time.

The second output graph shows the reaction force in the 1 ft dimension which is constraining the inside



axle to stay above the ground. We see that this force goes negative after the initial onset of cornering. This implies that the wheel would lose its grip on the road.

The above model had the rather unrealistic instantaneous application of an unbalanced



cornering force. A better model would have the force start at 0 and increase to 2486 over a finite period of time, perhaps 1 second. We can do this in Analytix by making the force a variable.

The variable time is updated during the simulation to contain the current simulation time. Hence we can create a time varying force by entering a formula which depends on the time variable.

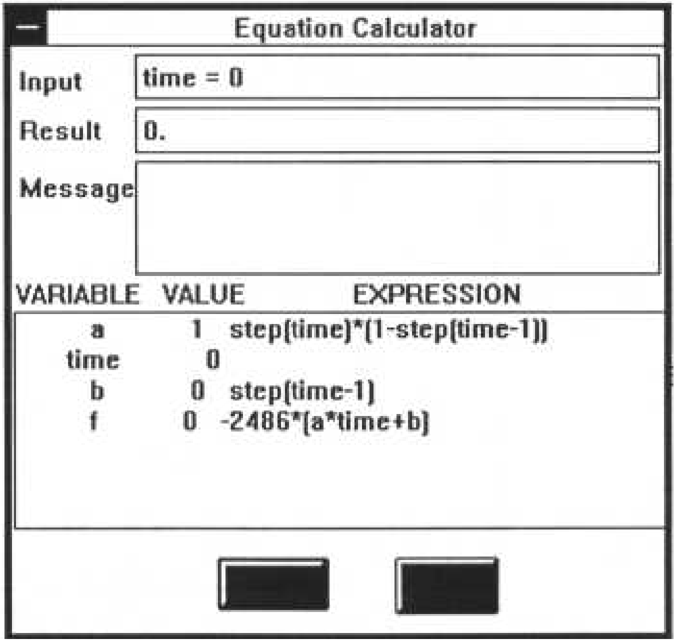
We will use a force which grows linearly from 0 to -2485 in the first second and stays at -2485 thereafter. We can build this function using step functions as follows:

a = step(time)*(1-step(time-1)

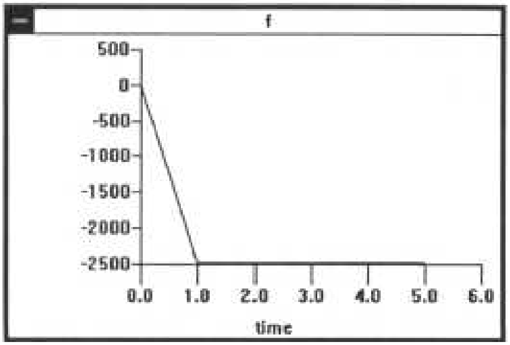
b = step(time-1)

f = -2486*(a*time+b)

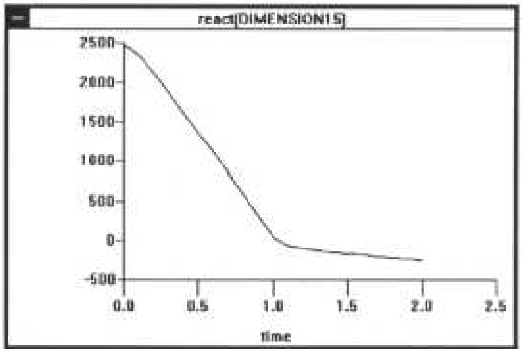
A graph of f against time verifies that this function is correct.



We can now enter f as the x component of the force and re-run our simulation. We see that, as we might



expect, the tire force is reduced more gradually, but still goes negative when the full side load is applied.



Dynamix

Menu Reference

Menu Reference

In this section, we provide reference documentation for the new menu options and dialog box selections which Dynamix adds to the Analytix package.

Use this reference manual in conjunction with the Analytix 2.0 manual.

Dimension Bounds

This facility lets you specify that a particular dimension is free to vary only between specified bounds.

Analysis / Add Stop

This option lets you specify impacts.

Analysis / Edit Stops

This option lets you change the coefficient of restitution for pre-specified impacts.

Analysis / Dynamic Simulation

This menu option lets you specify and perform a dynamic simulation.

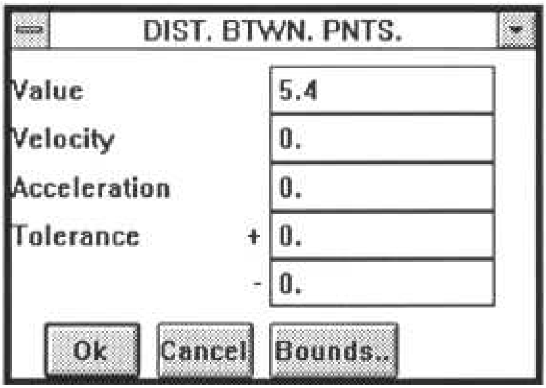
Dimension Bounds

It is sometimes useful in a dynamic simulation to be able to specify that a particular dimension is free to move within certain bounds.

For example a dimension which represents the length of a shock absorber may be free to move in response to an input force. However it may not extend beyond the maximum and minimum extensions of the shock absorber.

Another example is given in Dynamix Example 5, where a slot is modelled by a dimension which is free to move between bounds.

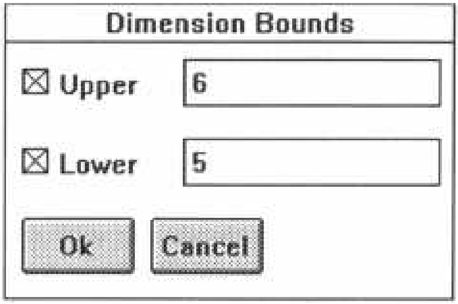
To model this type of behavior, Dynamix lets you give bounds for any dimension. To do this you click on the Bounds button in the Info Box for the dimension. (To get the Info Box for a dimension you just double click on the dimension.)



The Dimension Bounds Box now appears.

You can give the dimension an upper bound, a lower bound or both.

In a dynamic simulation, if this dimension is set free to move, then it will move dynamically so long as its value is less than any upper bound and greater than any lower bound. When the dimension reaches one



of the bounds, this will be modelled as a perfectly plastic collision with a restraint which enforces the dimension bound.

If the dimension is at its maximum value and under compression, then it will remain at its maximum value and not be free to move.

If the dimension is at its maximum value and under tension then it will be made free to move again.

If the dimension is at its minimum value and under tension, then it will remain at its minimum value and not be free to move.

If the dimension is at its minimum value and under compression then it will be made free to move again.

Analysis / Add Stop...

Stops are used to model collisions - situations where two components of a model collide and rebound.

A simple example of such a collision would be a ball bouncing on the ground.

A stop specifies the condition that a particular point will "stop against" a particular line. If the dynamic simulation determines that the point is about to pass through the line, it will not let this happen. Instead it will model the interaction between the point and the line as an instantaneous elastic/plastic collision. The point will bounce back away from the line with a speed determined by the coefficient of restitution given when specifying the stop.

A coefficient of restitution of 1 means that the collision will be modelled as perfectly elastic, and no energy is lost in the collision. A coefficient of restitution of 0 would be a perfectly plastic collision. A coefficient of restitution between 0 and 1 would imply some loss of energy during the impact.

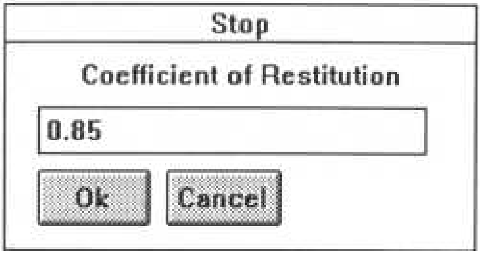
Note: due to the particular techniques used to integrate the dynamical equations, you shouldn't use stops to model totally plastic collisions, instead you should use bounds on a dimension.

To create a Stop you need to select the point and the line which will collide with each other.

First select the point then, holding down the Shift key select the line.

Now select Analysis / Add Stop.

You will see the Stop Dialog Box, which requires you to enter a coefficient of restitution. This should be a number greater than 0 and less than or equal to 1.



An alternative method of simulating collisions is to use a compression-only spring/damper combination. As the spring for such a collision would typically be very stiff, this introduces a technical condition called *stiffness* into the dynamics equations. Stiffness slows down the dynamics algorithms considerably and should be avoided if possible. Hence it is generally better to use stops rather than stiff springs.

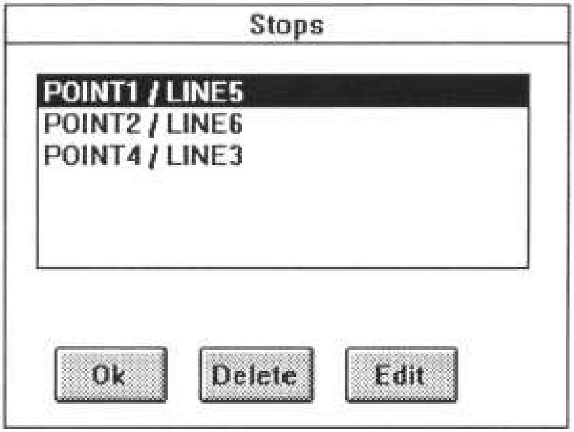
Values for the closing velocity and for the impulse at the collision may be examined by specifying that impulses be sent to the calculator when you start the dynamic simulation.

The impulse is a measure of force times time over the duration of the collision (assumed to be short.) The closing velocity is the component of the velocity of the point perpendicular to the line at impact.

Analysis / Edit Stops...

The Analysis / Edit Stops menu option lets you change the coefficient of restitution of Stops which you have already created.

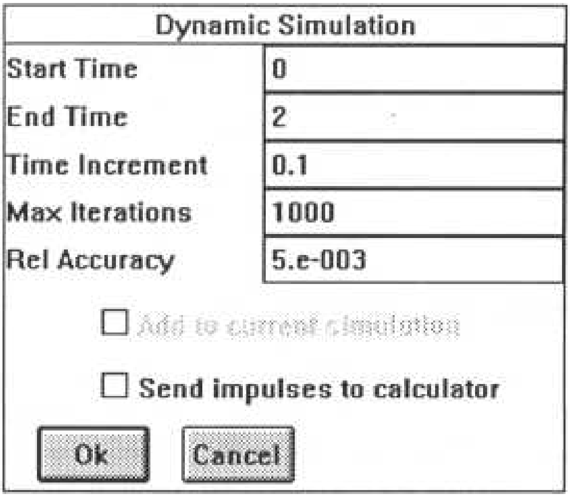
Alternatively, you can delete existing stops.



Analysis / Dynamic Simulation

To specify a dynamics simulation, you need to select the dimension or dimensions which will be free to move during the simulation. Then you should select Analysis / Dynamic Simulation.

You will see the Dynamic Simulation Dialog Box which lets you specify the time interval and accuracy of your simulation.



Time

It is necessary to specify the length of time over which you require the dynamic simulation to run. You specify a start time and an end time. You also specify a time interval at which you wish to record snapshots of the motion.

In the above example, the simulation will run for 2 seconds and a snapshot of the motion will be taken every 0.1 seconds.

It should be noted that this time interval is not the time step-size used by the integration routines. The integration routines calculate their own step size based on how complicated the behavior of the dynamical system is. The integration step size can be much smaller than the interval at which snapshots are saved.

During the integration, a special variable with the name *time is* kept updated at the current simulation time. You can use this variable to give a time varying force. For example you could set:

```
f=sin(2*pi*time)
```

and use f as the x component of an input force. this force would then vary sinusoidally with time in the simulation.

Setting a correct value for the variable *time* is why you need to enter a starting time and an end time for the simulation rather than just a duration.

Choosing a time interval

It is sometimes difficult to know what a sensible time interval for a specific dynamic simulation is.

If you have guessed a too long time interval, you will find that the simulation makes very slow (or even no) progress. This is because Dynamix has to do a large number of intermediate steps between each captured time step. In this situation, abort and start again with a smaller interval.

If the simulation is proceeding quickly, but the dimension values are not changing much from one interval to the next, you probably have too short a time interval. You can abort and start again with a longer interval.

It is sometimes helpful to do an initial inaccurate model just in order to determine an appropriate time interval. You make the model inaccurate and the simulation faster by setting the relative error high.

Max Iterations

The Maximum Iterations entry allows you to set a maximum for the number of integration time steps you will allow the system to perform. When the system determines it cannot perform the integration without using more than this number of steps, it will halt, giving you a warning message to that effect.

Note that the maximum iterations should be considerably greater than the number of time intervals stored. That is,

Max Iterations > (end time - start time)/time interval.

Relative Accuracy

The relative accuracy entry is the maximum allowable error in dimension values as a fraction of their values. A relative accuracy of 0.005 means that all dimension values throughout the simulation are accurate to within 0.5%.

Error is cumulative through the simulation. Hence the longer the total time interval set for the simulation, the harder it is to maintain a particular level of accuracy and the slower the system becomes.

To speed up the simulation, you can shorten the specified time interval or reduce the accuracy.

Add to Current Simulation

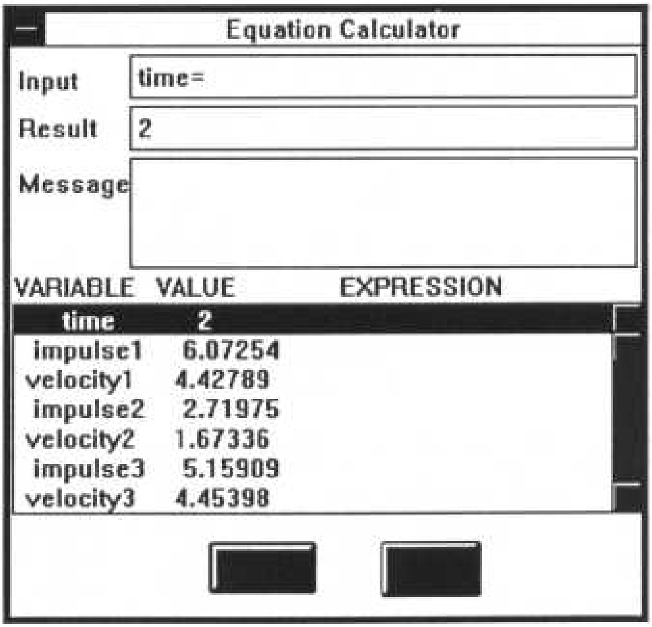
The Add to Current Simulation option lets you continue a previous simulation. The option is only available if a previous simulation exists.

When you select this option, the starting time for the simulation will be fixed to equal the ending time for the existing simulation. The time interval will be set to be the same as the interval for the previous simulation. The end time, maximum iterations and relative accuracy may all be reset.

Send Impulses to Calculator

If your model includes collisions (created by defining Stops between points and lines), then you can elect to send information about the collisions, if they occur, to the calculator.

The information which is sent is the value for the impulse and the closing velocity. For the first collision, the impulse value will be stored in the variable impulsel, the closing velocity in the variable velocityl.



Subsequent collisions will be recorded as *impulse2, velocity2, impulse3, velocity3,* etc.

You can inspect these values by opening up the Calculator Window by selecting Tools/Calculator.

The impulse is a measure of force times time and is the integral of the force exerted over the time of the collision The time of the collision is assumed to be infinitesimally short and the forces to be infinitely large.

The closing velocity is the relative velocity of the point and the line at impact.

Simulation

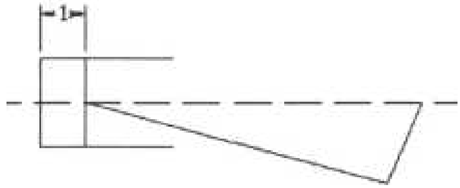
During the dynamic simulation, you will see an Abort box. Click on the Abort button to stop the simulation. The values which have already been calculated will be saved. You can resume computation by clicking on the Add to Current Simulation button of the Dynamic Simulation Dialog box.

You can work in another window while you are performing a dynamic simulation. However, the Dynamix window is inactive and so may not be reduced in size. In order to select another window, therefore, some portion must be visible before you start your simulation.

Which dimensions to use

When you create a dynamic model, most of the dimensions specify constraints which stay fixed throughout the simulation. One or more dimensions, however must be specified as free to move. There are usually a number of different ways to specify the free dimensions. The particular way you specify the free dimensions can affect the efficiency of the simualtion algorithms.

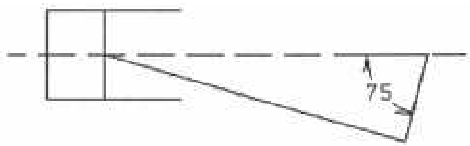
For example, a single cylinder engine may be modelled with the piston displacement as the free dimension, or it may be modelled with the crank



angle as the free dimension.

The piston displacement does not uniquely specify the configuration of the model, (the crank position could be up or down for any given piston displacement.) This presents a difficulty for the dynamics algorithms at the dead points at either end of the stroke.

On the other hand, using the crank angle as the free dimension in the mechanism uniquely defines it and



does not give the simulation any difficulty at the dead points.

A rule of thumb for deciding which free dimensions to use is to use angles in preference to distances and to use distance between point and line in preference to distance between two points or line length.

Outputs

The Graph, Table, Trace, Animate and Envelope tools all have the options to Use Current Simulation. With this option selected, these tools output results from the latest dynamic simulation.

For example to create an animation of the last simulation, select Tools/Animate and press the Use Current Simulation button.

To graph values of some output expression over the course of the simulation, select Tools/Graph, press the Use Current Simulation button and enter the desired expression as the y expression.

You can use the Analysis/Edit Simulation menu option to select a specific time instant in the simulation to use in the drawing. You may then inspect any of the output variables at that instant, or restart the simulation from there.