



Getting Started Using ADAMS/View

About This Guide	3
Introducing the Latch Design Problem	5
Building Your Model	11
Testing Your First Prototype	29
Validating Results Against Physical Test Data	41
Refining Your Design	49
Iterating Your Design	55
Optimizing Your Design	63
Automating the Design Process	71

U.S. Government Restricted Rights: If the Software and Documentation are provided in connection with a government contract, then they are provided with RESTRICTED RIGHTS. Use, duplication or disclosure is subject to restrictions stated in paragraph (c)(1)(ii) of the Rights in Technical Data and Computer Software clause at 252.227-7013. Mechanical Dynamics, Incorporated, 2300 Traverwood Drive, Ann Arbor, Michigan 48105.

The information in this document is furnished for informational use only, may be revised from time to time, and should not be construed as a commitment by Mechanical Dynamics, Incorporated. Mechanical Dynamics, Incorporated, assumes no responsibility or liability for any errors or inaccuracies that may appear in this document.

This document contains proprietary and copyrighted information. Mechanical Dynamics, Incorporated permits licensees of ADAMS[®] software products to print out or copy this document or portions thereof solely for internal use in connection with the licensed software. No part of this document may be copied for any other purpose or distributed or translated into any other language without the prior written permission of Mechanical Dynamics, Incorporated.

©2002 by Mechanical Dynamics, Incorporated. All rights reserved. Printed in the United States of America.

ADAMS[®] is a registered United States trademark of Mechanical Dynamics, Incorporated.
All other product names are trademarks of their respective companies.

About This Guide

Welcome to ADAMS/View

ADAMS/View is Mechanical Dynamics, Inc.'s (MDI's) powerful modeling and simulating environment. You can use ADAMS/View to build and simulate a model of any mechanical system that has moving parts. You can test your model in the same environments your actual product will experience. You can also validate your model against test data, and refine your model with flexible parts, realistic geometry, and controls. Finally, you can find the important parameters that affect your product design and find the right combination of design values to arrive at the best product design. Best of all, you can do all this over and over again, saving time and money.

Contents of This Guide

This guide provides overview information and step-by-step procedures for building, simulating, and refining a model using ADAMS/View. You will solve an actual design problem, as you construct a hand latch for closing large containers. In the process, you will learn how to use the features of ADAMS/View to solve your own design problems, from concept to finished product.

Additional tutorials are available online for add-on modules such as ADAMS/Flex, ADAMS/Controls, ADAMS/Hydraulics, and so on.

1

Introducing the Latch Design Problem

Overview

This tutorial introduces you to the process of solving an engineering design problem using ADAMS/View.

We assume that you will work through this tutorial in sequential order. Therefore, we give you more guidance in the beginning and less as you proceed through the tutorial. If you choose not to work through the tutorial in sequential order, you can import command files into ADAMS/View at different points in the tutorial and start working from there. You may, however, have to reference the beginning chapters for some of the basic concepts.



Look for the merge symbol at the beginning of each chapter to find out what file to import.

This chapter contains the following sections:

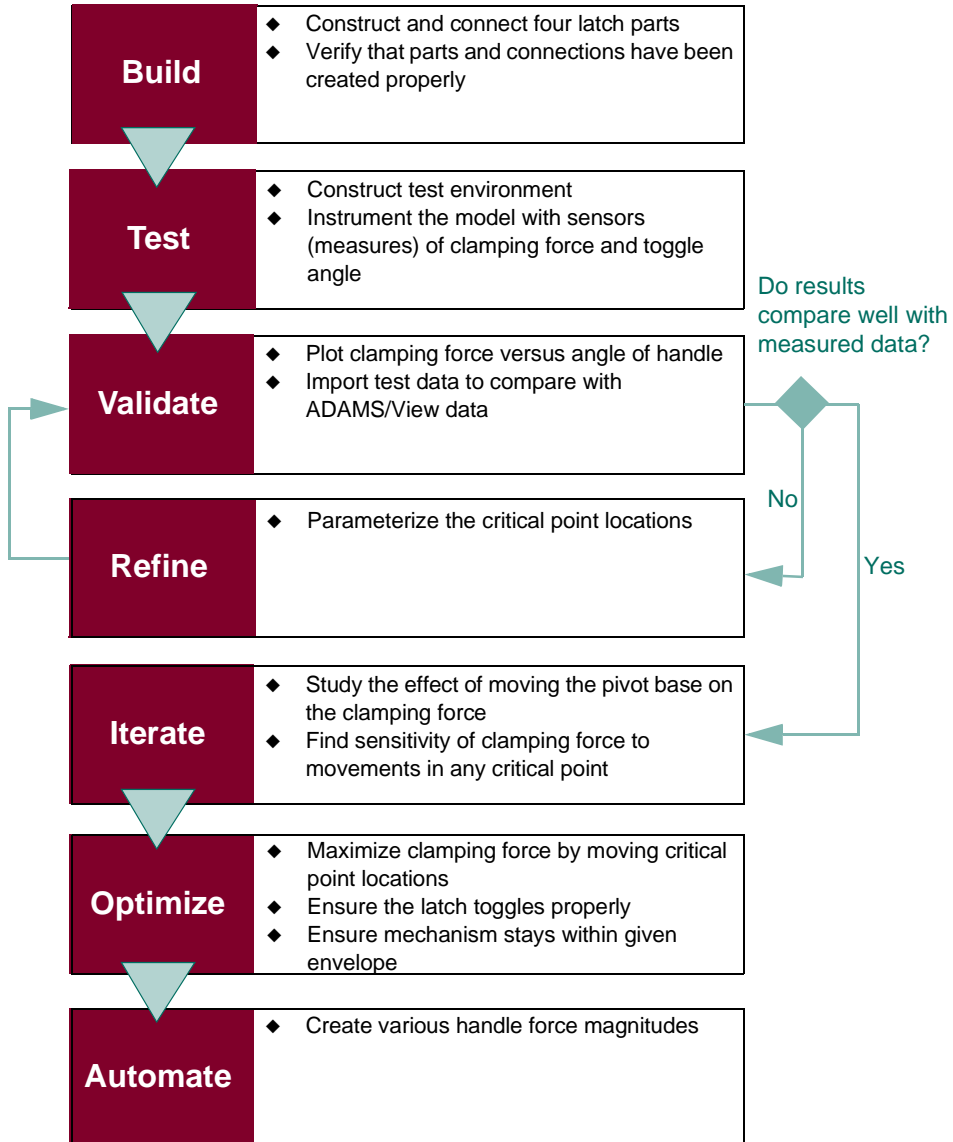
- [What You Will Learn, 6](#)
- [What You Will Create, 8](#)

What You Will Learn

The tutorial leads you through the design process steps outlined in [Figure 1](#) on page 7. These are the seven basic steps you should follow whenever you use ADAMS/View to create and test models:

- 1** Build a model of a mechanical system that includes moving parts, joints, flexible connections, and applied forces.
- 2** Test your design by simulating the motion of the model as it would perform in actual operation.
- 3** Validate your design by comparing physical test data with the results of the virtual simulation.
- 4** Refine your model so your virtual test data matches your physical test data.
- 5** Iterate your design to assess the sensitivity of its performance to various design changes.
- 6** Optimize your design to find the best combination of design parameters to achieve maximum performance.
- 7** Automate the design process steps so you can quickly try out various design options.

Figure 1. Design Process Steps for the Latch Model

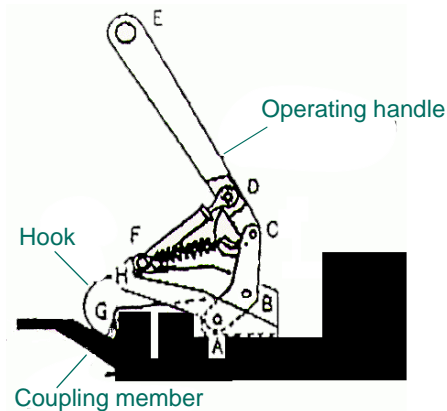


What You Will Create

In this tutorial, you'll build a latch model that is required to securely clamp two halves of large shipping containers together. The latch design problem is based on a latch developed by Earl V. Holman of North American Aviation, Inc., Downey, CA, under contract to the Manned Spacecraft Center in Houston. This latch is one of twelve used to clamp the Lunar Excursion Module to the Command and Service Module in the Apollo program.

Figure 2 shows an illustration of the physical model of the latch.

Figure 2. Physical Model of Hand Latch Design



Design Requirements

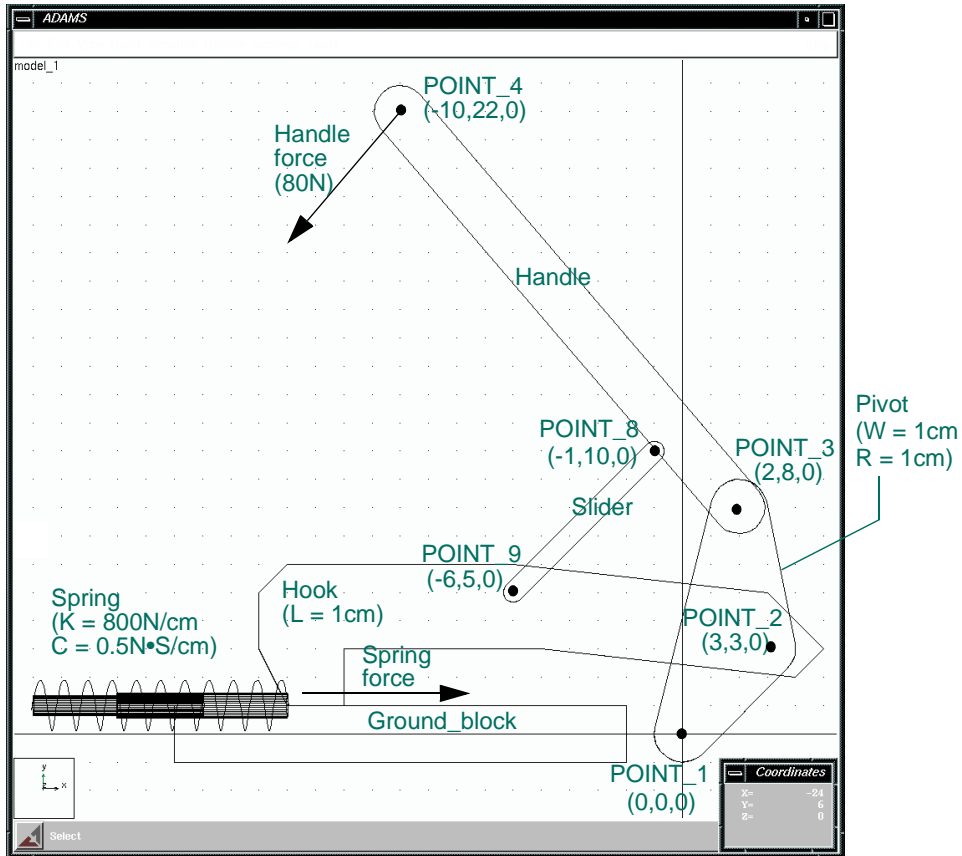
The latch model must meet the following design requirements:

- Exerts at least 800 N clamping force.
- Is hand-actuated by less than 80 N force.
- Is hand released with minimal effort.
- Must work within a given envelope.
- Clamping remains secure under vibration.

How the Latch Model Works

Figure 3 on page 10 shows a virtual model of the latch. The latch is clamped by pushing down on the operating handle at POINT_4. This causes the pivot to rotate around POINT_1 in a clockwise direction, drawing back POINT_2 of the hook. As this happens, POINT_8 of the slider is forced downward. Finally, as POINT_8 passes through the line between POINT_9 and POINT_3, the clamping force reaches its maximum. POINT_8 should move below the line created by POINT_3 and POINT_9, followed by the operating handle coming to rest on the top of the hook. This sets the latch near the maximum force point, but allows a reasonable release force to open the latch.

Figure 3. ADAMS/View Latch Model



Based on the description of the latch operation, the relative layout of POINT_1 through POINT_9 is important in ensuring that the latch will meet the design requirements. Therefore, when your latch model is assembled and tested, you will want to change the relative locations of the points to see their effect on the design requirements.

2

Building Your Model

Overview

In this chapter you build and connect the parts of the toggle latch. You also verify that you've created the parts and connections properly. Once you have a valid model, you will proceed to test it in a realistic situation, as explained in [Testing Your First Prototype](#) on page 29.

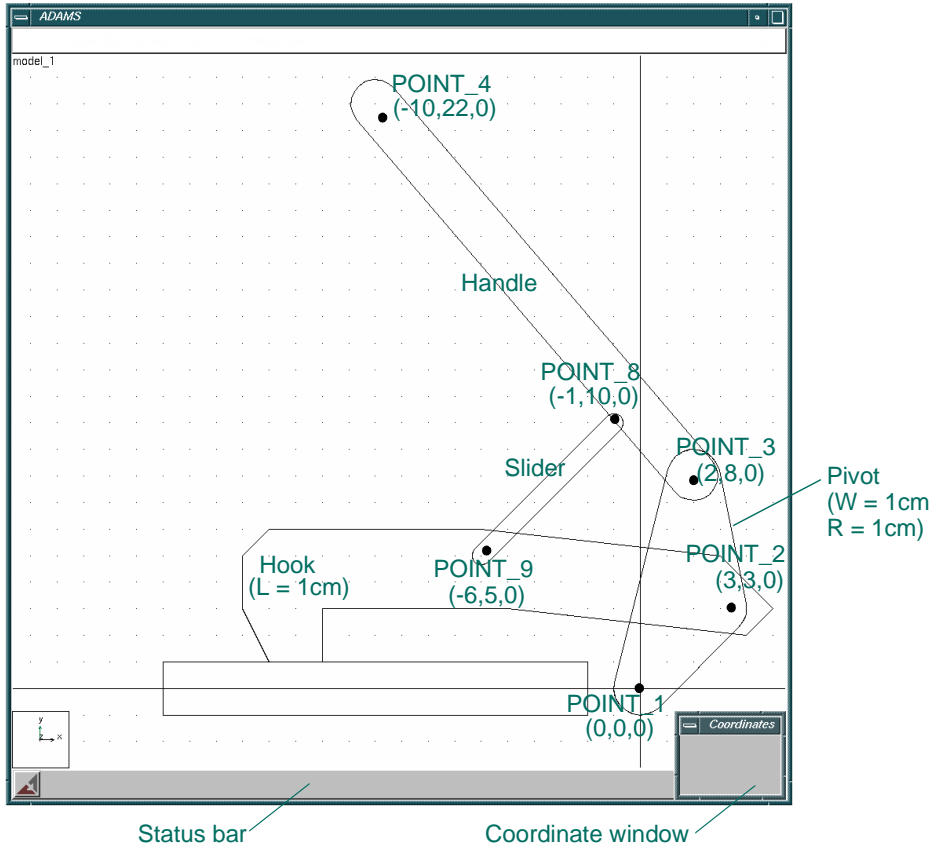
ADAMS/View assigns to the parts you create the material type of steel, with a material density of 7801.0 kg/m^3 .

You build the latch model in two basic sections:

- [Building the Pivot and Handle, 13](#)
- [Building the Hook and Slider, 24](#)

[Figure 4](#) on page 12 shows the latch as it should look when you have finished it.

Figure 4. Latch in Build Phase



Building the Pivot and Handle

As the initial steps in building the latch model, you perform the following tasks:

- Starting ADAMS/View and Creating a Database, 13
- Familiarizing Yourself with ADAMS/View, 15
- Setting Up Your Work Environment, 17
- Creating Design Points, 18
- Creating the Pivot, 19
- Renaming the Pivot, 20
- Creating the Handle, 21
- Connecting the Parts Using Revolute Joints, 21
- Simulating the Motion of Your Model, 22
- Seeing the Effect of Parameterization, 23

Starting ADAMS/View and Creating a Database

In this section, you start ADAMS/View and create a modeling database containing a new model named Latch. A modeling database contains all your work in the current session of ADAMS/View. It contains any models you create, their attributes, simulation results, plots, customized menus and dialog boxes, and any preferences you set.

In the UNIX environment you start ADAMS/View from the ADAMS Toolbar. For information on the Toolbar, see the guide, [Running and Configuring ADAMS on UNIX](#).

In the Windows environment you start ADAMS/View from the Start button. For more information, see the guide, [Running ADAMS on Windows](#).

To start ADAMS/View in the UNIX environment:

- 1 At the command prompt, enter the command to start the ADAMS Toolbar, and then press **Enter**. The standard command that MDI provides is `adamsx`, where `x` is the version number, for example `adams12`.

The ADAMS Toolbar appears.

- 2 Select the **ADAMS/View** tool .

The Welcome dialog box appears on top of the ADAMS/View main window.

To start ADAMS/View in the Windows environment:

- 1 Select **Start**.
- 2 Point to **Programs**, point to **ADAMS 12.0**, point to **AView**, and then select **ADAMS - View**.

The Welcome dialog box appears.

To create a database from the Welcome dialog box:

- 1 Select **Create a new model**.
- 2 Replace the contents of the **Model name** text box with **Latch**.
- 3 Select **OK**.


Familiarizing Yourself with ADAMS/View

Before continuing with the tutorial, familiarize yourself with the ADAMS/View interface. In particular, familiarize yourself with the Main Toolbox, shown at the right.

You use the Main Toolbox to access all geometric construction elements, including joints and forces. Some of the tools in the Main Toolbox are part of tool stacks. These tools have a small triangle in the lower right corner. The default tool or the last selected tool appears on top of the tool stack. To see the tools contained in each tool stack, refer to the ADAMS/View Main Toolbox quick card included with this guide.

If ADAMS/View does not automatically display the Main Toolbox, you can display it as explained next.

To display the Main Toolbox:

- In the lower right corner of the main ADAMS/View window, select the **Toolbox** tool .

The Main Toolbox appears on the left side of the screen.

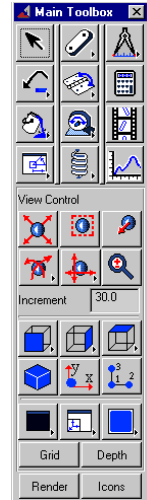
To open tool stacks and select a tool:

- 1 In the Main Toolbox, right-click on one of the tools that has a small triangle in the lower right corner, as shown in the first illustration in [Figure 5](#) on page 16.

A tool stack appears.

- 2 Move the cursor over the tool you want to use, and click.

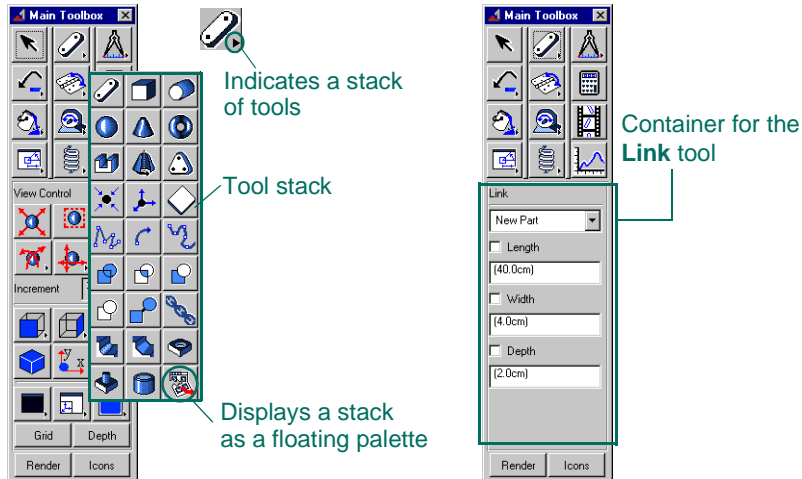
The bottom half of the Main Toolbox, the container, changes depending on the tool or tool stack that you select, as shown in the second illustration in [Figure 5](#) on page 16.



To return to the default configuration:

- Select the Select tool .

Figure 5. Main Toolbox and Associated Elements



Learning About Getting Help

When working in ADAMS/View, you can get help in a number of ways:

- Use the online guides to read overviews, procedures, and see many examples. From the **Help** menu, select **ADAMS/View Guides**.
- Use the online help to learn about entering values in a dialog box or using an ADAMS/View tool. While working in a dialog box, press the **F1** key.

For more information about getting help in ADAMS/View, see the printed *Quick Reference Guide* that comes with your license.

Setting Up Your Work Environment

In this section you set your units, specify the grid size, and display the coordinate window. You can change the units at any time during the modeling process, even while reading and writing model or results data files. You can use the ADAMS/View working grid and the coordinate window to establish map points and receive feedback on precise locations for the design layout.

To set up your work environment:

- 1 From the **Settings** menu, select **Units**.

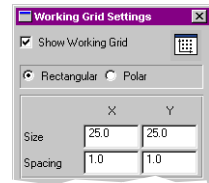
The Units Settings dialog box appears.

- 2 Set the units of length to centimeter.
- 3 Select **OK**.
- 4 From the **Settings** menu, select **Working Grid**.

The Working Grid Settings dialog box appears.

- 5 Set the grid size along X and Y to 25, and the grid spacing at 1.

Note: You do not need to enter units. However, because you set the units to centimeters earlier, ensure that you do not enter mm after the values in the **Size** and **Spacing** text boxes.



- 6 Select **OK**.
- 7 From the **Settings** menu, select **Icons**.

The Icon Settings dialog box appears.

- 8 In the **New Size** text box, enter 1.5.

Note: The text box **New Size** is colored gray. Throughout ADAMS/View gray text boxes means that the text box is optional and you do not have to enter a value to execute the command.

- 9 Select **OK**.

To display the Coordinates window:


- 1 From the **View** menu, select **Coordinate Window**.
The Coordinates window appears.
- 2 Move your cursor around the screen and notice the coordinate location labels that appear next to the cursor.

Creating Design Points

ADAMS/View enables you to change the layout of your design to quickly find the best mechanism for the application. You use points to lay out your design so that you can make layout changes by moving the points. Points are used to define locations in space on which you can position and parameterize other objects. Points are the easiest way to parameterize the geometry of a model because they let you specify important locations once and build other modeling objects from them. Parameterization becomes important later on when you work on refining your latch model.


To create design points:



- 1 From the Main Toolbox, select the **Select** tool, and then select the **Dynamic Pick** tool  to zoom in on your grid. To learn how to use the Main Toolbox, see [Familiarizing Yourself with ADAMS/View](#) on page 15.

- 2 Drag the cursor across the area of the working grid display.



- 3 Right-click the **Rigid Body** tool stack (**Link** tool is on top by default) to display the tool stack containing the **Point** tool.
- 4 Select the **Point** tool  and click the locations shown in [Table 1](#) on page 19 to place design points. Use the default settings for point, which are **Add to Ground** and **Don't Attach**.

Note: To create multiple points without having to select the **Point** tool again, double-click the tool. To deselect the **Point** tool, select the **Select** tool.

Table 1. Points Coordinate Locations


	X location:	Y location:	Z location:
POINT_1	0	0	0
POINT_2	3	3	0
POINT_3	2	8	0
POINT_4	-10	22	0

Creating the Pivot

Create the pivot using the **Plate** tool. The pivot represents the pivot part, as shown in [Figure 4](#) on page 12.

To create the pivot:



- 1 Select the **Plate** tool .
- 2 In the **Thickness** text box, enter 1, and then press **Enter**.

Note: Pressing **Enter** every time you enter text into a text box ensures that ADAMS/View accepts the entered value.

- 3 In the **Radius** text box, enter 1, and then press **Enter**.
- 4 Click the location of POINT_1, POINT_2, and POINT_3.

Note: The ADAMS/View status bar provides instructions on how to perform operations. When you move your cursor on the screen during an operation, the status bar displays what you should do next. The status bar is located below the ADAMS/View main window. For the location of the status bar, see [Figure 4](#) on page 12.

- 5 To close the plate geometry, right-click.

Renaming the Pivot

As you create objects, ADAMS/View automatically assigns names to them. Each name consists of a string containing the object type and a unique integer ID for that type. For example, ADAMS/View named the recently created part PART_2.

The full name of an object is comprised of the concatenated names of its parent similar to a directory structure in a file system. Because PART_2 is located within the model named Latch, its full name is .Latch.PART_2, while the full name of the marker named cm on PART_2 is .Latch.PART_2.cm.

In this section, you rename PART_2 to pivot. ADAMS/View understands that you are just renaming the part, so the model name Latch remains.

To rename the pivot:

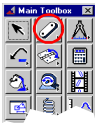
- 1 Right-click the plate part.
A pop-up menu appears.
- 2 Point to **Part: PART_2**, and then select **Rename**.
The Rename Object dialog box appears.
- 3 Replace **PART_2** with **pivot**, as shown in the following dialog box:



Creating the Handle

You can create the handle using the Link tool.

To create the handle:



1 Select the Link tool .

2 Click POINT_3, then POINT_4 to create a link between the two points.

Note: Only attach the link to a point when that point label is visible.

3 Rename the link part, Part: PART_3, to **handle**, to represent the handle part as shown in Figure 4 on page 12.

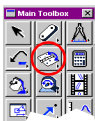
Connecting the Parts Using Revolute Joints

A revolute joint is an attachment between two parts that allows one part to move with respect to another part about a common axis.

In this section, you place a revolute joint between the pivot and ground, which will allow the pivot to rotate with respect to ground. You also place a second revolute joint between the pivot and handle to allow the two parts to rotate with respect to each other.

You will use the construction method **2 Bod - 1 Loc**, and **Normal To Grid** to create the revolute joints, which is the default method. When you use this method for creating revolute joints you select the two bodies to attach and then select the location.

To connect the parts using revolute joints:

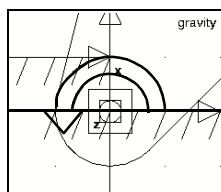


1 Select the Revolute Joint tool .

2 To select the parts to attach, click the **pivot** and **ground** (the background).

3 Click POINT_1 to set the joint's location.

The revolute joint at POINT_1 should look like this:



Note: If you did not change the icon size as explained in Step 7 on page 17, you will not see the revolute joint on the screen. Go back and change the icon size as explained in Step 7. In the Icon Settings dialog box, be careful not to accidentally turn off the display of icons.

- 4 Select the **Revolute Joint** tool again.
- 5 Select the **pivot**, the **handle**, and **POINT_3**.




Simulating the Motion of Your Model

In this section you set the simulation parameters, and simulate the motion of the model to see if you've assembled the parts and joints properly. You set the simulation end time and the number of output steps to tell ADAMS/View how long you want the simulation to run, and the frequency with which you want it to output the data.

During the simulation, the handle moves in a circular motion with respect to the pivot, as the pivot moves in a circular motion with respect to ground. Note that gravity is acting on the model.

To simulate the motion of your model:



- 1 Select the **Simulation** tool .
- 2 Set up a simulation with an end time of 1 second and 50 steps.
- 3 Select the **Simulation Start** tool .
The model simulates, and then remains in simulate mode.
- 4 To return to the initial model configuration, select the **Reset** tool .

Seeing the Effect of Parameterization

Since you parameterized your latch model using points, you can move a point and the related objects update automatically. For example, if you move POINT_1, the pivot and the joint move along with it since they were created on top of that point.

To see the effect of parameterization:

- 1 Right-click on POINT_1.

A pop-up menu appears.

- 2 Point to **Point: POINT_1**, and then select **Modify**.

The Table Editor appears, listing the points in your model and their locations.

- 3 Set the x location for POINT_1 to -2.

- 4 Select **Apply**.

POINT_1, along with the joint and the pivot, move to the new location.

Note: You can move the Table Editor out of the way by clicking and dragging its top window border.

- 5 Set the x location for POINT_1 back to 0.

- 6 Select **OK**.

Building the Hook and Slider

The last steps in building your latch model are:


- [Creating the Hook and the Slider, 24](#)
- [Connecting the Parts Using Revolute Joints, 26](#)
- [Simulating Your Model, 27](#)
- [Saving Your Database, 27](#)

Creating the Hook and the Slider

You now create the hook using the Extrusion tool. An extrusion is a three-dimensional object defined by its profile and depth. To create an extrusion, you draw a polyline that defines the extrusion's profile. By default, ADAMS/View extrudes the profile along a line perpendicular to the working grid.

To create the hook:



- 1 Select the **Extrusion** tool .
- 2 Be sure that **Create profile by:** is set to **Points** and **Closed** is selected.
- 3 Select **Path:** to **About Center**.
- 4 In the **Length** text box, enter 1, and then press **Enter**.
- 5 Click the locations listed in [Table 2](#) on page 25.


- 6 To close the extrusion, right-click.

Small boxes, called hot points, appear at the vertices of the extrusion.

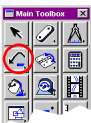
Note: Sometimes ADAMS/View snaps to the nearest object instead of snapping to a coordinate value. To override this, hold down the **Ctrl** key and move the cursor until you select the desired coordinate.

Table 2. Extrusion Coordinate Values

X location:	Y location:	Z location:
5	3	0
3	5	0
-6	6	0
-14	6	0
-15	5	0
-15	3	0
-14	1	0
-12	1	0
-12	3	0
-5	3	0
4	2	0

Note: If you make a mistake while creating the extrusion, continue creating the extrusion, and then, at the end, use the extrusion hot points to place the points in the correct locations. You can also use the Undo tool  to cancel any of the changes you made. You can undo up to 10 actions.

- 7 Change the name of the extrusion part to **hook**.



To create the slider:

- 1 Create two more design points, POINT_8 and POINT_9, at the locations shown in Table 3.

Tip: Sometimes ADAMS/View snaps to the nearest object instead of snapping to a coordinate value. To override this, hold down the Ctrl key and move the cursor until you select the desired coordinate.

Table 3. Points Coordinate Locations

	X location:	Y location:	Z location:
POINT_8	-1	10	0
POINT_9	-6	5	0

- 2 Create a link connecting these two new design points. Again, before you click, make sure the point labels are visible.
- 3 Rename the link part to **slider**.

Connecting the Parts Using Revolute Joints

You now create three more revolute joints.

- Place revolute joints at the following locations using the construction method **2 Bod - 1 Loc**, and **Normal To Grid**:
 - Between the handle and the slider at POINT_8.
 - Between the slider and the hook at POINT_9.
 - Between the hook and the pivot at POINT_2.

Tip: You can see the names of objects that are in the same location by right-clicking on the objects. A pop-up menu appears with the names of all the objects at that location. This is also helpful if you have forgotten the names of objects.

Simulating Your Model

In this section you simulate again to see if you've assembled the parts and joints properly. During this simulation, the handle, the hook, and the slider move in a circular motion with respect to the pivot, as the pivot moves in a circular motion with respect to ground. The only force acting on this model is gravity, since you have not assigned any forces to it.

To simulate the motion of your model:

- 1 Select the **Simulation** tool.
- 2 Run a 1 second, 50-step simulation.
- 3 Select the **Reset** tool to return to the initial model configuration.

Saving Your Database

Use the Save Database As command to save the current modeling database as an ADAMS/View binary file. Saving your modeling database as a binary file saves all modeling information.

To save your database:

- From the **File** menu, select **Save Database As**, and then save the file as **build**.

3

Testing Your First Prototype

Overview

In this chapter, you prepare the latch model for virtual testing, and then proceed to test it. Virtual tests allow you to quickly set up and tear down tests in the virtual environment.

In the test phase you'll be doing the following:

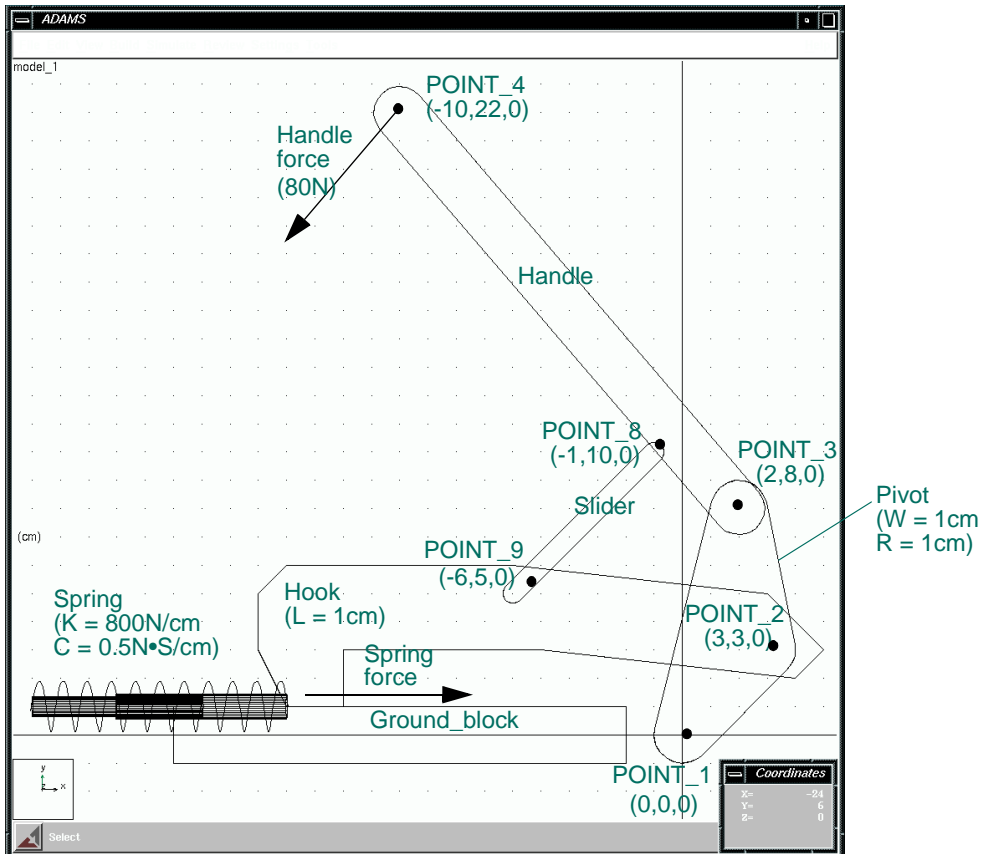
- Creating the Ground Block, 31
- Adding a Three-Dimensional Contact, 31
- Adding a Spring, 33
- Creating a Handle Force, 34
- Creating a Measure on the Spring Force, 35
- Creating an Angle Measure, 36
- Creating a Sensor, 38
- Saving Your Model, 39
- Simulating Your Model, 39



If you choose not to work through the tutorial in sequential order and want to start here, import the file `build.cmd` from `install_dir/aview/examples/Latch/`, where `install_dir` is the directory where ADAMS is installed.

Figure 6 shows how your model should look at the end of the test phase.

Figure 6. Latch Model in Test Phase




Creating the Ground Block

You use the **Box** tool to create a ground block. The ground block represents the surface on which the hook slides. By default, ADAMS/View creates the block with a depth that is twice the length of the shortest dimension of the block. You can also specify the length, height, and depth of the block before you draw it.

To create the ground_block:




- 1 Select the **Box** tool , and change its construction method from **New Part** to **On Ground**.
- 2 Click at location (-2, 1, 0) and drag to (-18, -1, 0). Alternatively, you can click at the start location and then click again at the end location.
- 3 Rename the part ground to **ground_block**.

Adding a Three-Dimensional Contact

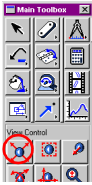
In this section, you add a three-dimensional contact between the hook and **ground_block**. The three-dimensional contact models how hook and **ground_block** interact when the latch closes.


To add a three-dimensional contact:



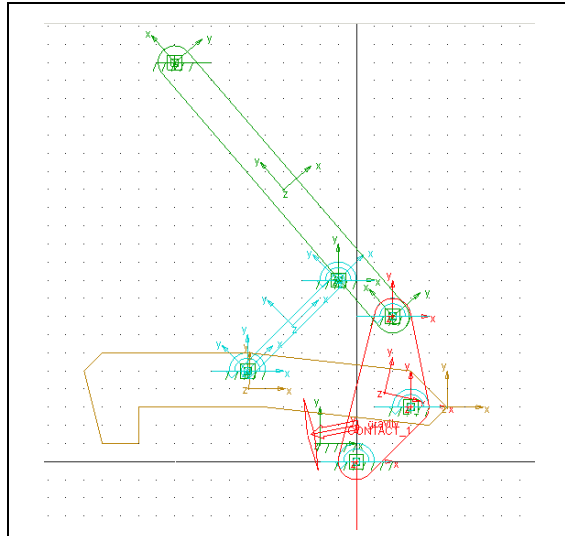
- 1 Use the **Dynamic Pick** tool to zoom in on the area around the hook end.
- 2 From the **Force (Connector)** toolstack, select the **Contact** tool .

The Create Contact dialog box appears.
- 3 Right-click the **First Solid** text box, point to **Contact_Solid**, and then select **Pick**. Select the **hook** (EXTRUSION_7).
- 4 Now do the same for the **Second Solid** text box, selecting the **ground_block** (BOX_11).
- 5 Because you will use the default values for the contact force, select **OK**.



- 6 Select the **Select** tool, and then select the **Fit** tool .

ADAMS/View fits your entire model into the current window. It should look similar to the one shown here:





Adding a Spring

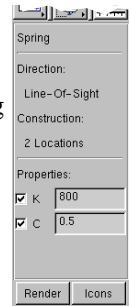
The spring between the hook and ground represents the clamping force when the hook engages the container. The spring should have a stiffness coefficient of 800 N/cm and a damping coefficient of $0.5 \frac{\text{N} \cdot \text{s}}{\text{cm}}$, so that a hook movement of 1.0 cm gives a large clamping force.

As you create the spring, ADAMS/View automatically creates markers to define the location and orientation of the spring. Because the hook vertex is not in the plane of the model, after you create the spring, you should modify the marker at the vertex so the spring is in the plane of the model.

To add a spring:



- 1 Use the **Dynamic Translate** tool  to move your model to the right, to give yourself room to add the spring.
- 2 Select the **Translational Spring-Damper** tool  to create a spring between the ground and the hook.
- 3 In the Spring container of the Main Toolbox, select the toggle for spring stiffness coefficient, **K**, and for damping coefficient, **C**.
- 4 Set **K** to **800** and **C** to **0.5**.
- 5 To add the spring, click at the following locations:
 - ◆ (-14, 1, 0) (make sure the hook vertex, .HOOK.EXTRUSION_7.V16, is selected, not the coordinate location).
 - ◆ (-23, 1, 0)



A red spring appears.


To modify a marker:

- 1 Right-click the location (-14, 1, 0).
A pop-up menu appears.
- 2 Point to **Marker: MARKER_24**, and then select **Modify**.
The Marker Modify dialog box appears.
- 3 In the **Location** text box, change the location to (-14.0, 1.0, 0.0).
ADAMS/View puts the spring in the plane of the model.
- 4 Select **OK**.
- 5 Use the **View Fit** tool to fit your model to the screen.

Creating a Handle Force

In this section you create a handle force with a magnitude of 80N, representing a reasonable force to be applied by hand.

To create a handle force:

- 1 Select the **Force (Single-Component)** tool  and do the following in the Force container on the Main Toolbox:
 - ◆ Set the Run-time Direction to **Space Fixed**.
 - ◆ Set the **Characteristic** to **Constant**.
 - ◆ Select **Force**, and then set it to **80**.
- 2 Select the following in the order listed:
 - ◆ The handle
 - ◆ A marker near the handle end point
 - ◆ The location -18, 14, 0

ADAMS/View creates a handle force.



Creating a Measure on the Spring Force

ADAMS/View uses measures to monitor important model characteristics during simulation. To give you visual feedback on the progress of the simulation during interactive simulations, ADAMS/View displays the measure values.

For the latch model, you need a measurement of the clamping force when the latch is clamped so you can compare it with the design requirement. The spring force measure represents the clamping force.

To create a measure on the spring force:

- 1 Right-click the spring, point to **Spring: SPRING_1**, and then select **Measure**.

The AssemblyMeasure dialog box appears.

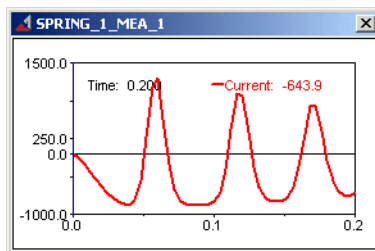
- 2 Set **Characteristic** to **force**.

- 3 Select **OK**.

The spring measure strip chart appears.

- 4 Run a 0.2 second, 50-step simulation.

A graph of the clamping force appears during the simulation, as shown next:



- 5 Select the **Reset** tool to return to the initial model configuration.

Creating an Angle Measure

Now you create an angle measure to ensure that the handle pushes down far enough to engage the latch. When the latch engages, you know that the handle has toggled “overcenter” and is now capable of holding the latch in a secure position. It’s similar to using vice grips--the point where the vice clamps down on the material is the toggle point.

Creating the angle measure involves picking markers located at the points defining the angle, in this case POINT_8, POINT_3, and POINT_9. ADAMS/View automatically defines markers at the center of mass of all parts and at vertices and endpoints of geometry. ADAMS/View also creates markers automatically when you constrain objects, like when you add a joint between parts. Therefore, there can be several markers at one location. When you create the angle measure you only need to pick one marker for each location. However, you must make sure that the marker you pick belongs to the appropriate part.

To create an angle measure:

- 1 From the **Build** menu, point to **Measure**, point to **Angle**, and then select **New**.
The Angle Measure dialog box appears.
- 2 In the **Measure Name** text box, enter the measure name as **overcenter_angle**.
- 3 Right-click the **First Marker** text box, point to **Marker**, and then select **Pick**.
- 4 Pick the markers to enter in your measure as shown in row 1 of [Table 4](#) on page 37 and illustrated in [Figure 7](#) on page 37.

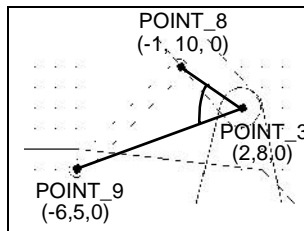
Note: Do not pick markers that belong to ground, because markers used in the angle measure must move with the latch.

- 5 Repeat the above two steps for the Middle Marker and Last Marker.

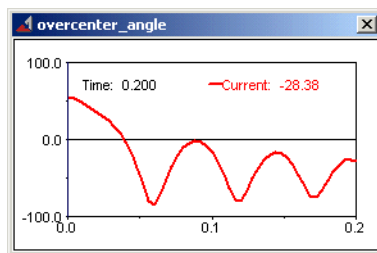
Table 4. overcenter_angle Measure Markers

Angle points:	Marker location:	Coordinate values:
First Point	Any marker at POINT_8	-1, 10, 0
Middle Point	Any marker at POINT_3 (angle vertex)	2, 8, 0
Last Point	Any marker at POINT_9	-6, 5, 0

Figure 7. Graphical Representation of overcenter_angle



- 6 Select OK to display your angle measure strip chart as shown next:

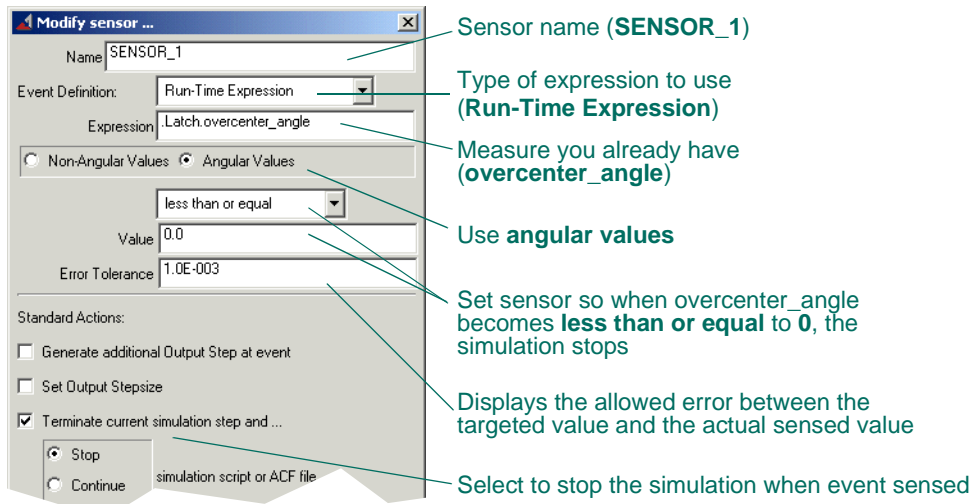


Creating a Sensor

You now create a sensor to detect when `overcenter_angle` goes below zero, meaning that the latch has toggled properly. When this condition is met, the sensor automatically stops the simulation.

To create a sensor:

- 1 From the **Simulate** menu, point to **Sensor**, and then select **New**.
The Create Sensor dialog box appears.
- 2 Modify the Create Sensor dialog box as shown next, and then select **OK**:



Note: If you get an error when trying to create the sensor, type in a new name for the sensor, press **Enter**, and then continue filling in the dialog box as shown above.

Saving Your Model

Save your latch model to record your progress.

To save your model:

- From the File menu, select **Save Database As**, and then save this file as test.

Simulating Your Model

Simulate your model to see if it is assembled properly and that the sensor works to stop the simulation when the overcenter_angle measure becomes less than or equal to 0, indicating that the latch has reached its toggle point.

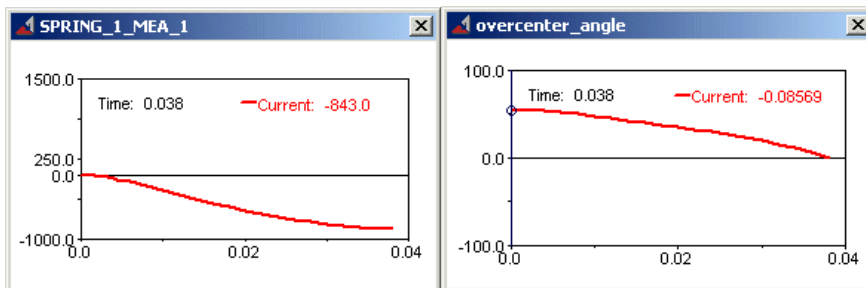
To simulate your model:

- 1 Select the **Simulation** tool and run a 0.2-second simulation with 100 steps.

As the simulation proceeds, ADAMS/View updates the strip charts for the spring force and angle measures to show that the sensor stopped the simulation.

A message window also appears alerting you that ADAMS/View stopped the simulation because of the sensor.

These strip charts show that ADAMS/View stopped the simulation as the latch reached the toggle point:



- 2 Select the **Reset** tool to return to the initial model configuration.

4

Validating Results Against Physical Test Data

Overview

In this chapter, you compare physical test data with virtual test data. By comparing the two sets of data, you immediately know the limitations of your model compared to the physical prototype, and you will have all the data in one place to be able to eliminate the differences.

Validating against your physical test data involves:

- Importing Physical Test Data, 42
- Creating a Plot Using Physical Test Data, 43
- Modifying Your Plot Layout, 45
- Creating a Plot Using Virtual Test Data
- Saving Your Model, 47



If you choose not to work through the tutorial in sequential order and want to work through it starting here, import the file `test.cmd` from `install_dir/aview/examples/Latch/`, where `install_dir` is the directory where ADAMS is installed. After importing `test.cmd`, simulate your model as shown on page 39.

Importing Physical Test Data

In this section you import physical test data to compare with your virtual test data. We created the physical test data by setting the `overcenter_angle` on a physical model and measuring the clamping force. ADAMS/View takes the physical test data and creates two measures. One measure, `MEA_1`, contains the x data and the other measure, `MEA_2`, contains the y data for your plot.

To import physical test data:

- 1 From the **File** menu, select **Import**.
The File Import dialog box appears.
- 2 Set the **File Type** to **Test Data**.
- 3 Make sure that the **Create Measures** option is selected.
- 4 In the **File to Read** text box, enter: `install_dir\aview/examples/Latch/test_dat.csv`, where `install_dir` is the directory where ADAMS is installed.
- 5 In the text box to the right of the **Model Name** menu, enter `.Latch`.
- 6 Select **OK**.

Note: Because this is test data, nothing appears on the screen when you import the data.

Creating a Plot Using Physical Test Data

You can now take the two measures created by your physical data, and use them to create a plot for comparison.

To create a plot using physical test data:

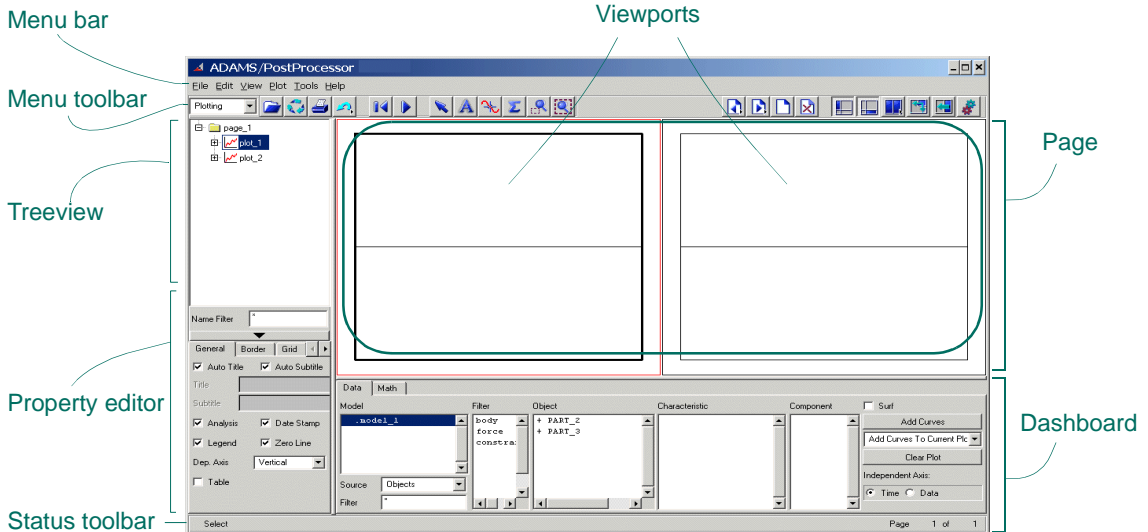
- 1 On the Review menu, select Postprocessing or press F8.

ADAMS/View launches ADAMS/PostProcessor (see [Figure 8](#)), a post-processing tool that lets you view the results of simulations you performed.

ADAMS/PostProcessor has three modes: plotting (default), animation, and report, as shown in the first option menu on the menu toolbar.

Take a minute to familiarize yourself with ADAMS/PostProcessor. For more information, see the guide, [Using ADAMS/PostProcessor](#).

Figure 8. ADAMS/PostProcessor

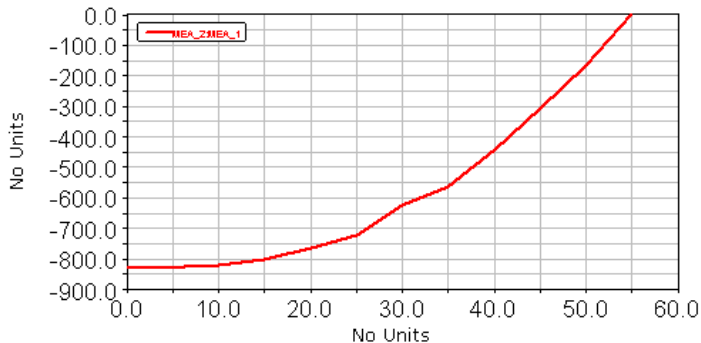


- 2 At the bottom left of the dashboard, set **Source** to **Measures**.
- 3 From the **Simulation** list, select **test_dat**.
- 4 At the bottom right of the dashboard, set **Independent Axis** to **Data**.

A browser, named Independent Axis Browser, appears. It lets you select data for the horizontal axis.

- 5 Select **MEA_1**.
- 6 Select **OK**.
- 7 From the dashboard, from the **Measure** list, select **MEA_2**, for the vertical axis data.
- 8 Select **Add Curves** to add the new data to the plot.

ADAMS displays the plot of the two measures as shown next:



Modifying Your Plot Layout

To make your plot ready for a report, you assign a title to the plot, assign units to the x- and y-axis, and modify the plot legend.

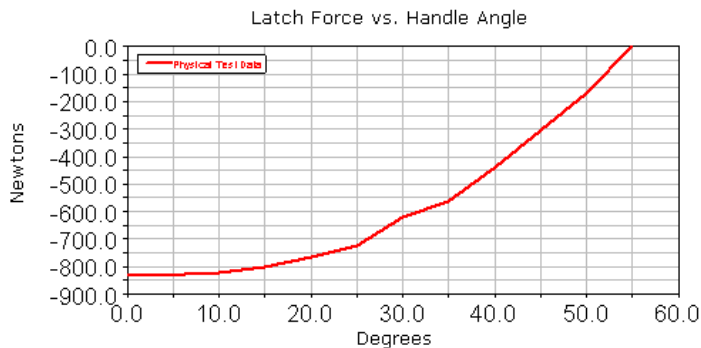
To assign units:

- 1 From the treeview, double-click **page_1**.
- 2 Select **plot_1**.
- 3 In the Property Editor, under the tab **General**, clear the selection of **Auto Title**.
- 4 In the **Title** text box, enter **Latch Force vs. Handle Angle**.
- 5 Press **Enter**.
- 6 From the treeview, double-click **plot_1**.
- 7 From the treeview, select **haxis**.
- 8 In the Property Editor, select the tab **Labels**.
- 9 In the **Label** text box, enter **Degrees**, and then press **Enter**.
- 10 Repeat the procedure for **vaxis**, labeling it **Newtons**.

To modify the plot legend:

- 1 From the treeview, select **curve_1**.
- 2 In the **Legend** text box, enter **Physical Test Data**.

Your plot should look similar to the one shown next:



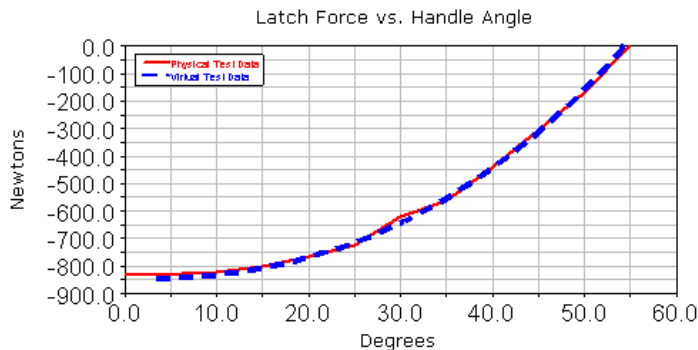
Creating a Plot Using Virtual Test Data

Use the `overcenter_angle` and the `SPRING_1_MEA_1` measures to create a plot of your virtual test data, to use in your comparison. After you examine your plots, you will see that the physical test data is not exactly the same as the virtual test data. The data is in close agreement, however.

You cannot use the curve math tools to subtract the two curves since the physical test data has different independent data than the virtual test data.

To create a plot using virtual test data:

- 1 In the dashboard, from the **Simulation** list, select **Last_Run (...)**.
- 2 Set **Independent Axis** to **Data**.
The **Independent Axis Browser** appears.
- 3 Select `overcenter_angle`, for the horizontal axis data.
- 4 Select **OK**.
- 5 From the **Measure** list, select `SPRING_1_MEA_1`, for the vertical axis data.
- 6 Select **Add Curves**.
- 7 Change the legend text for this curve to **Virtual Test Data**.



- 8 From the **File** menu, select **Close Plot Window**.

Saving Your Model

To save your model:

- From the File menu, select **Save Database As**, and then save the file as **validate**.

5

Refining Your Design

Overview

In this chapter you refine your model to add more parametrics to the critical point locations. This allows you to compare different layouts of the model to the clamping force.

To add more parametrics you will perform two tasks:

- [Creating Design Variables, 50](#)
- [Reviewing Design Variable Values, 52](#)



If you choose not to work through the tutorial in sequential order and want to work through it starting here, import the file `test.cmd` from `install_dir/aview/examples/Latch/`, where `install_dir` is the directory where ADAMS is installed.

Creating Design Variables

You now create design variables to represent the design points in your model, except for POINT_4, the handle end point. You do not modify POINT_4, because doing so would not meet envelope requirements for the mechanism. In subsequent chapters you will use design variables to run design studies and optimization.

To create design variables:

- 1 Right-click the design point POINT_1 (0, 0, 0), point to **Point: POINT_1**, and then select **Modify**.

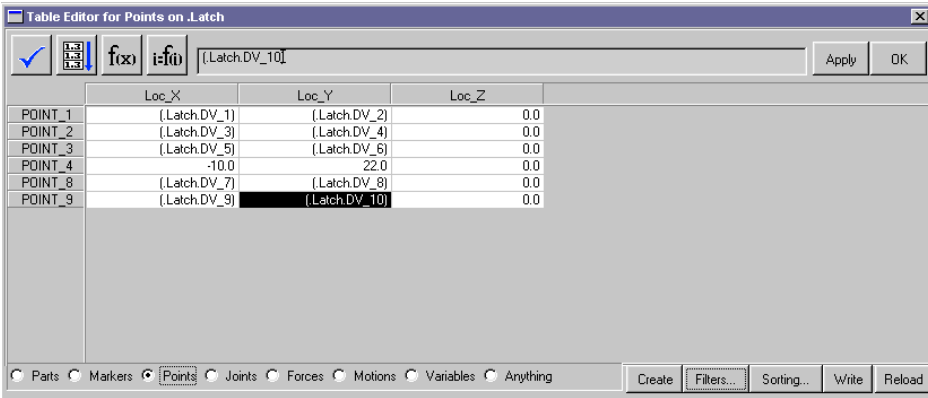
The Table Editor appears.

- 2 Select the **Loc_X** cell for POINT_1.
- 3 Right-click the input box at the top of the Table Editor, point to **Parameterize**, point to **Create Design Variable**, and then select **Real**.

This creates a design variable named **.Latch.DV_1** with the value of 0.

- 4 Select the **Loc_Y** cell for POINT_1.
- 5 Repeat Step 3.

- 6 Repeat the above procedure for the x and y locations of POINT_2, POINT_3, POINT_8, and POINT_9. Your Table Editor dialog box should look like the one shown here:



	Loc_X	Loc_Y	Loc_Z
POINT_1	{Latch.DV_1}	{Latch.DV_2}	0.0
POINT_2	{Latch.DV_3}	{Latch.DV_4}	0.0
POINT_3	{Latch.DV_5}	{Latch.DV_6}	0.0
POINT_4	-10.0	22.0	0.0
POINT_8	{Latch.DV_7}	{Latch.DV_8}	0.0
POINT_9	{Latch.DV_9}	{Latch.DV_10}	0.0

Note: Make sure your design variable names map to the design point locations, as shown above. This becomes important later on as you perform your design study and optimization.

- 7 Select **Apply**.

Design variables now control the locations of your design points, with the exception of the POINT_4 location.

Reviewing Design Variable Values

After you've created all the design variables, you can display their range and allowed values. ADAMS/View sets the design variable range based on the envelope requirements for the latch. It automatically assigns a +/- 10% relative range to the design variables, except when the design variable real value is 0. When the design variable value is 0, the range is set as +/- 1 absolute.

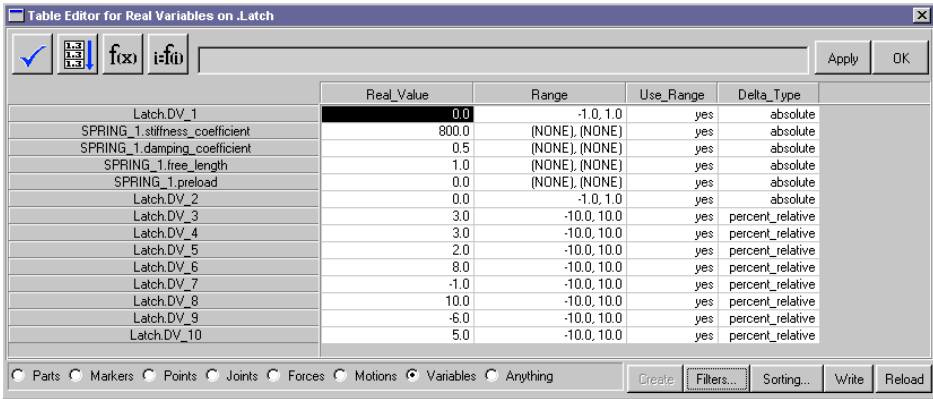
If you want to open up the range to different values, you must modify the range values and possibly the delta type (which you will do in [Chapter 6](#)). This restricts the layout changes to your model, to keep it within a specified envelope.

To review design variable values:

- 1 At the bottom of the Table Editor, select the **Variables** option.
- 2 Select **Filters**.
The Variables Table Editor Filters dialog box appears.
- 3 Select **Delta Type**.
The delta type lets you know if the range value is absolute or percent relative.
- 4 Be sure that **Range** is selected.

5 Select OK.

The Table Editor changes to show you the range of the design variables.



	Real_Value	Range	Use_Range	Delta_Type
Latch.DV_1	0.0	-1.0, 1.0	yes	absolute
SPRING_1.stiffness_coefficient	800.0	(NONE), (NONE)	yes	absolute
SPRING_1.damping_coefficient	0.5	(NONE), (NONE)	yes	absolute
SPRING_1.free_length	1.0	(NONE), (NONE)	yes	absolute
SPRING_1.preload	0.0	(NONE), (NONE)	yes	absolute
Latch.DV_2	0.0	-1.0, 1.0	yes	absolute
Latch.DV_3	3.0	-10.0, 10.0	yes	percent_relative
Latch.DV_4	3.0	-10.0, 10.0	yes	percent_relative
Latch.DV_5	2.0	-10.0, 10.0	yes	percent_relative
Latch.DV_6	8.0	-10.0, 10.0	yes	percent_relative
Latch.DV_7	-1.0	-10.0, 10.0	yes	percent_relative
Latch.DV_8	10.0	-10.0, 10.0	yes	percent_relative
Latch.DV_9	-6.0	-10.0, 10.0	yes	percent_relative
Latch.DV_10	5.0	-10.0, 10.0	yes	percent_relative

6 Select OK.

7 Save your model as refine.

6

Iterating Your Design

Overview

In this chapter, you work on arriving at an improved design that meets the specifications and includes all necessary behavior of the physical latch. You set up some design studies for a few points to find a case that maximizes peak clamping force, while making sure the handle toggles overcenter.

In the process of iterating your design you will be:

- [Performing a Manual Study, 56](#)
- [Running a Design Study, 57](#)
- [Examining the Results of Design Studies, 61](#)



If you choose not to work through the tutorial in sequential order and want to work through it starting here, import the file `refine.cmd` from `install_dir/aview/examples/Latch/`, where `install_dir` is the directory where ADAMS is installed.

Performing a Manual Study

As you perform this manual study, you will see how your spring force increases as you manually modify the design variable DV_1. If the spring force strip chart is not displayed, you should display it before modifying your design variable.

To update your spring force strip chart:

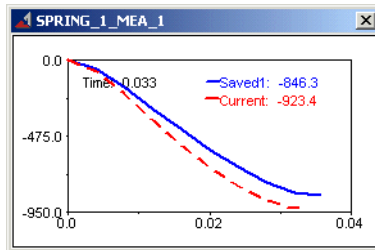
- 1 From the **Build** menu, point to **Measure**, and then select **Display**.
The Database Navigator appears.
- 2 Select **SPRING_1_MEA_1**.
- 3 Select **OK**.
- 4 Run a .2 second, 100-step simulation and then return to the initial model configuration.
ADAMS/View updates the spring measure strip chart.
- 5 Right-click the spring force curve in the strip chart. Point to **Curve: Current**, and then select **Save Curve**.

To modify a design variable:

- 1 From the **Build** menu, point to **Design Variable**, and select **Modify**.
The Database Navigator appears.
- 2 Double-click on **DV_1**.
The Modify Design Variable dialog box appears.
- 3 Change the standard value of **DV_1** to 1.0.
- 4 Select **OK**.

- 5 Run a .2 second, 100-step simulation.

This new plot shows a comparison of the spring force measure for the two cases. The new curve shows better draw on the spring.



- 6 Change DV_1 back to 0.0, its original value.

Running a Design Study

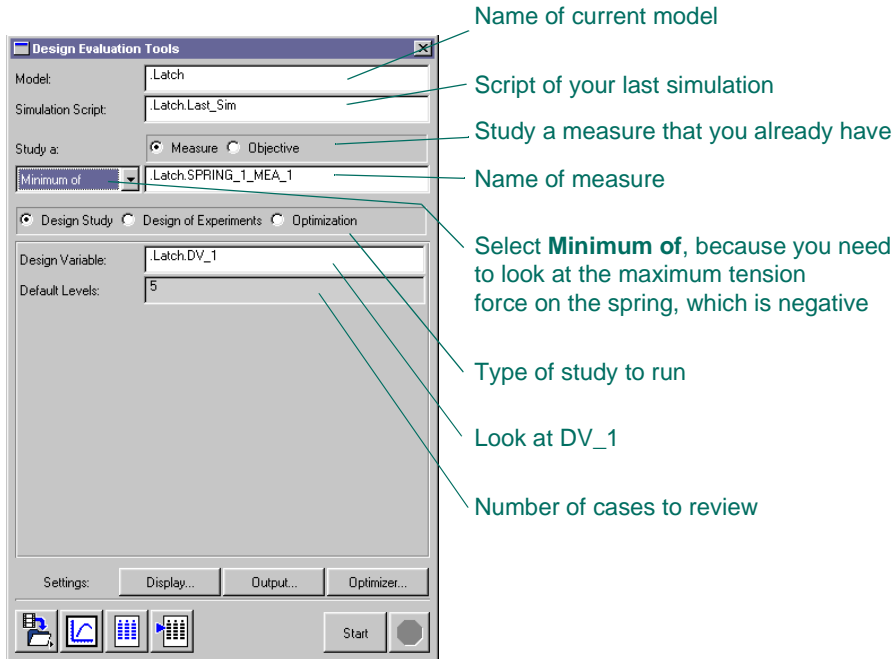
Run a design study to quickly look at a range of design variable values, and see how they affect the design. ADAMS/View gives you the option of displaying various plots, as well as a design study report. The design study report includes the design sensitivity of the pivot location, which is the rate of change of the spring force with respect to a change in the pivot location.

To run a design study:

- 1 From the **Simulate** menu, select **Design Evaluation**.

The Design Evaluation Tools dialog box appears.

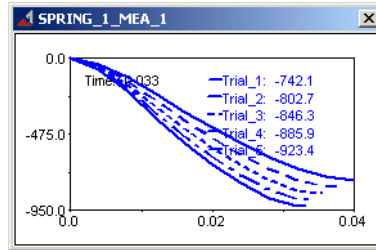
- Fill out the dialog box that appears so it matches the one shown here. Leave the dialog box open.



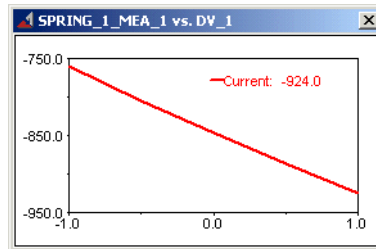
- In the Design Evaluation Tools dialog box, select **Display**.
The Solver Settings dialog box appears.
- To see all the options, select **More**.
- Set **Time Delay** to 0.0.
- Set **Chart Objective**, **Chart Variables**, and **Show Report** to **Yes**.
- In the Solver Settings dialog box, select **Close**.
- In the Design Evaluation Tools dialog box, select **Start**.

ADAMS displays the following plots and a design study report:

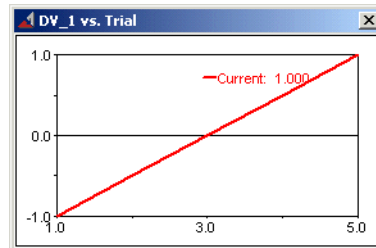
- Spring force plot, with five curves for the five different cases:



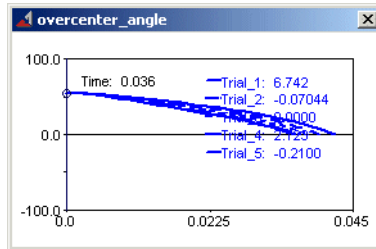
- SPRING_1_force versus DV_1 plot:



- DV_1 versus Trial plot:



- overcenter_angle plot, with five curves for the five different cases:



- Design study report:

Information

Apply Parent Children Modify Verbose Clear Read from File Save to File Close

Design Study Summary

Model Name : Latch
Date Run : 2000-01-20 14:14:34

Objectives

01) Minimum of SPRINC_1_MEA_1
Units : NO UNITS
Maximum Value: -760.208 (trial 1)
Minimum Value: -924.261 (trial 5)

Design Variables

V1) DV_1
Units : NO UNITS

Trial	01	DV_1	Sensitivity
1	-760.21	-1.0000	-88.070
2	-804.24	-0.50000	-86.058
3	-846.27	0.00000	-82.111
4	-886.35	0.50000	-77.994
5	-924.26	1.0000	-75.812

Note: Your results might vary slightly (about 1%) from those shown here due to computational differences.

- 9 Close the Information window, the Message window, and the Design Evaluation Tools dialog box.

Examining the Results of Design Studies

To save you time, we ran some design studies for all the design variables and provided the results in Table 5. These results are from individual design studies of each design variable, keeping the rest of the variables fixed at their nominal value.

The results help you determine which design variables you should use for your optimization study, because they represent a summary of the sensitivity of the clamping force magnitude to a given change in the geometric location, keeping all other locations fixed.

We found sensitivities by using the plot statistics function and finding the slope of the design study curve for spring force plotted against design variable value.

Table 5. Design Studies Results

ADAMS design variable names:	Design point locations:	Initial value:	Sensitivity at initial value (N/cm):	Apparent optimal:
DV_1	(POINT x)	0	-82	1
DV_2	(POINT y)	0	56	0
DV_3	(POINT_2 x)	3	142	2.7
DV_4	(POINT_2 y)	3	-440	3.3
DV_5	(POINT_3 x)	2	-23	2.2
DV_6	(POINT_3 y)	8	281	7.6
DV_7	(POINT_5 x)	-1	36	-1.1
DV_8	(POINT_5 y)	10	-287	10.5
DV_9	(POINT_6 x)	-6	-61	-5.4
DV_10	(POINT_6 y)	5	104	4.5

Parameterization lets you see which design variables have the greatest effect on the clamping force. In this case, design variables DV_4, DV_6, and DV_8 have the greatest sensitivity.



7

Optimizing Your Design

Overview

You will now work on finding an optimal design that best meets the performance parameters, while satisfying the design constraints. Optimization involves determining which objective function you want to minimize or maximize, selecting the design variables you want to change, and specifying constraint functions that must be satisfied.

Optimizing your design involves:

- [Modifying Design Variables, 64](#)
- [Running an Optimization, 66](#)



If you choose not to work through the tutorial in sequential order and want to work through it starting here, import the file `refine.cmd` from `install_dir/aview/examples/Latch/`, where `install_dir` is the directory where ADAMS is installed.

Modifying Design Variables

Use the design of experiments results shown in [Table 5](#) (those with the largest variable sensitivities) to determine which variables to use in optimization. Set up the optimization on design variables DV_4, DV_6, and DV_8, since they seem to have the most effect on the clamping force. Run the optimization with the set of parameters that would maximize spring force.

To make sure that the model stays within a given work envelope, set the following limits on the design variables:

Table 6. Design Variable Limits

Design variable name:	Design point location:	Minimum value:	Maximum value:
DV_4	POINT_2 y	1	6
DV_6	POINT_3 y	6.5	10
DV_8	POINT_5 y	9	11

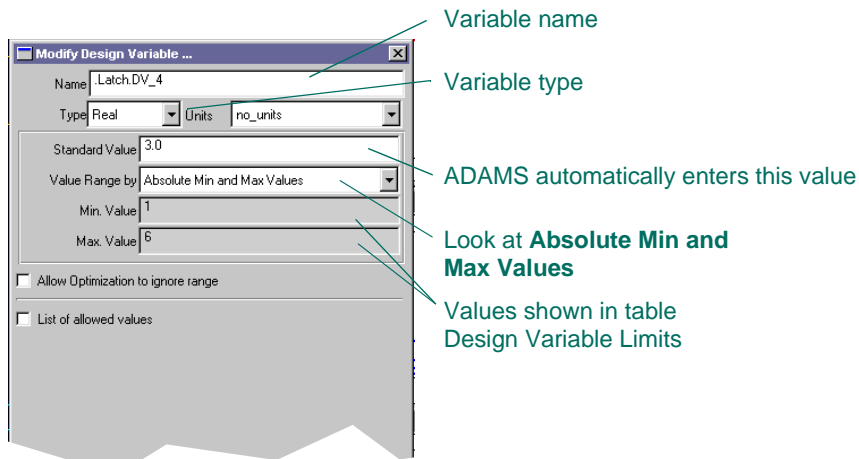
To modify design variables:

- 1 On the **Build** menu, point to **Design Variable**, and then select **Modify**.

The Database Navigator appears.

- 2 Double-click the first design variable you need to modify, in this case, **DV_4**.

The Modify Design Variable dialog box appears:



- 3 Set the **Min. Value** and **Max. Value**, which are the minimum and maximum values for your first design variable, DV_4, as shown in [Table 6](#) on page 64.
- 4 Select **Apply**.
- 5 Right-click the **Name** text box, point to **Variable**, and **Browse** for DV_6.
- 6 Double-click on DV_6.
- 7 Type in the minimum and maximum values for DV_6. Make sure the **Absolute Min and Max Values** option is selected.
- 8 Select **Apply**.
- 9 Repeat the above three steps for DV_8.
- 10 After you've modified the last design variable, DV_8, select **OK**.

Running an Optimization

At this point you'll run an optimization to find the best design configuration. Before you run an optimization, display the spring force measure. If you imported refine.cmd, the spring force measure will already be displayed.

To display the spring force measure:

- 1 On the **Build** menu, point to **Measure**, and then select **Display**.
The Database Navigator appears.
- 2 Select **SPRING_1_MEA_1**.
- 3 Select **OK**.
The **SPRING_1_MEA_1** plot appears.

To display the overcenter_angle:

- 1 On the **Build** menu, point to **Measure**, and then select **Display**.
The Database Navigator appears.
- 2 Select **overcenter_angle**.
- 3 Select **OK**.
- 4 The **overcenter_angle** plot appears.

To run an optimization:

- 1 On the **Simulate** menu, select **Design Evaluation**.
The Design Evaluation Tools dialog box appears.

2 Fill in the dialog boxes as shown below and select **Start** after selecting **Close** from the two Solver Settings dialog boxes.

The screenshot shows the 'Design Evaluation Tools...' dialog box. Annotations include:

- 'Name of current model' pointing to the 'Model' field containing '.Latch'.
- 'Script of your last simulation' pointing to the 'Simulation Script' field containing '.Latch_Last_Sim'.
- 'Use **Measure**, since you already have one' pointing to the 'Measure' radio button under 'Study a:'.
- 'Measure name' pointing to the 'Minimum of' dropdown menu containing '_Latch.SPRING_1_MEA_1'.
- 'Select **Minimum of**, because you need to look at the maximum tension force on the spring, which is negative' pointing to the 'Minimum of' dropdown.
- 'Type of study to run' pointing to the 'Optimization' radio button.
- 'List of variables to be changed in optimization (DV_4, DV_6, and DV_8)' pointing to the 'Design Variables' list.
- 'Find optimal clamping force by minimizing the spring force (largest negative value)' pointing to the 'Goal' dropdown menu containing 'Minimize Des. Meas. / Objective'.

Be sure to set this to **Database Storage**

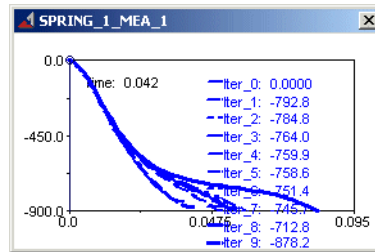
Select **More** to see all options

The screenshot shows two 'Solver Settings' dialog boxes. The left one has 'Output' selected in the 'Category' dropdown, and the 'Output Category' is set to 'Database Storage'. The right one has 'Optimization' selected in the 'Category' dropdown. Both have 'More' checked at the bottom. Annotations include:

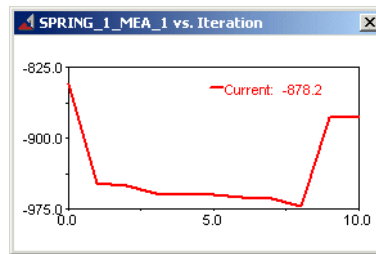
- An arrow pointing from the 'Database Storage' annotation to the 'Output Category' dropdown in the left dialog.
- An arrow pointing from the 'More' annotation to the 'More' checkbox in the left dialog.
- An arrow pointing from the 'More' annotation to the 'More' checkbox in the right dialog.

Note: The optimization may take several minutes to complete, and will take several iterations. Look in the status bar for status information.

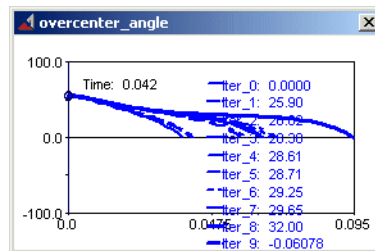
The spring force measure plot shows the optimal clamping force as a function of time.




The SPRING_1_force versus iteration plot shows how the spring force changed with each iteration.



The overcenter_angle plot shows the cases in which the angle reached the toggle point.



To see the final variable values and the optimized clamping force value:

- 1 At the bottom of the Design Evaluation Tools dialog box, select the **Tabular Report** tool .

The Design Evaluation Results Table dialog box appears.

- 2 In the **Result Set** text box, enter `.Latch.Multi_Run_001.Optimization_Results`.
- 3 Select **OK**.

In the Information window that appears, you see all the data obtained from the optimization, including the initial and final values of the spring force and DV_4, DV_6, and DV_8.

8

Automating the Design Process

Overview

In this chapter you work on meeting the last two design requirements for the latch: hand-actuated by less than 80.0 N force and hand released by less than 5.0 N of force. To do so, you quickly and interactively try out various handle force magnitudes by:

- [Creating Design Variables, 72](#)
- [Creating a Custom Dialog Box, 72](#)
- [Modifying Force Values, 77](#)



If you choose not to work through the tutorial in sequential order and would like to work through it starting here, import the file `optimize.cmd` from `install_dir/aview/examples/Latch/`, where `install_dir` is the directory where ADAMS is installed.

Creating Design Variables

To find the handle force values, you need to create two new design variables, DV_11 and DV_12. Use DV_11 to set the clamping force, and DV_12 to set the release force.

To create design variables:

- 1 From the **Build** menu, point to **Design Variable**, and then select **New**.
The Create Design Variable dialog box appears.
- 2 Set **Standard Value** at **80**.
- 3 Set **Value Range by** to **Absolute Min and Max Values**.
- 4 Set the minimum value at **60**, and the maximum at **90**.
- 5 Select **Apply**.
- 6 Repeat Steps 2 through 4 for DV_12. Set the standard value to **10**, the minimum value to **0**, and the maximum to **20**.
- 7 Select **OK**.

Creating a Custom Dialog Box

You now create a custom dialog box to use in adjusting the downward force and the release force. You do this by:

- [Adding Objects to Your Dialog Box, 73](#)
- [Assigning Commands to the Sliders, 75](#)
- [Testing the Dialog Box, 76](#)
- [Saving Your Dialog Box, 76](#)

To create a custom dialog box:

- 1 From the **Tools** menu, point to **Dialog Box**, and then select **Create**.
The Dialog-Box Builder appears.
- 2 From the **Dialog Box** menu, select **New**.
The New Dialog Box appears.
- 3 In the **Name** text box, enter **Force_Control**.
- 4 From the **Create Buttons** option, select **OK** and **Close**.
- 5 Select **OK**.
ADAMS/View displays your custom dialog box:

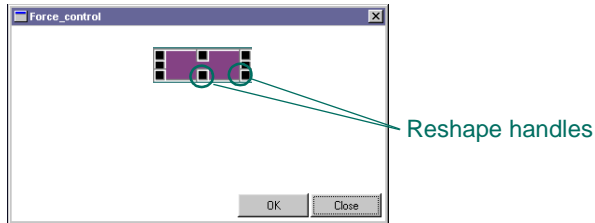


Adding Objects to Your Dialog Box

In this section you add two sliders to your dialog box and you label the sliders. The sliders let you set a value from a continuous range of possible values, as detailed in the next section.

To add sliders to your dialog box:

- 1 Activate the Dialog-Box Builder.
- 2 From the Dialog-Box Builder **Create** menu, select **Slider**.
- 3 Inside your dialog box, click near the top center area of your dialog box, which is where the slider should appear, as shown in [Figure 9](#) on page 74.
- 4 Double-click on the slider to select it.
- 5 Drag the reshape handles to resize the slider.

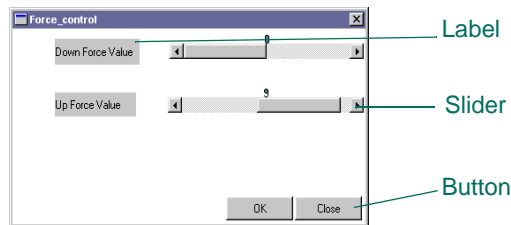


- 6 Create another slider underneath the first one.
- 7 Resize this slider to match the size of the first slider.

To label the sliders:

- 1 From the Dialog-Box Builder **Create** menu, select **Label**.
- 2 Inside your dialog box, click to the left of the first slider, which is where the label should appear, as shown in [Figure 9](#).
- 3 Double-click on the label, to select it.
- 4 Set the Dialog-Box Builder **Attributes** option menu to **Appearance**.
- 5 Replace the contents of the **Label Text** text box, with **Down Force Value**.
- 6 Select **Apply** to make the change in your dialog box.
- 7 Use the reshape handles to resize your label.
- 8 Create another label, **Up Force Value**, to the left of the second slider.

Figure 9. Force Control Dialog Box



Assigning Commands to the Sliders

You can assign commands to the sliders you just created to perform the functions you need.

To assign commands to the sliders:

- 1 Double-click the Down Force Value slider, and then from the Dialog-Box Builder **Attributes** option menu, select **Commands**.

- 2 At the bottom of the Dialog-Box Builder, select **Execute commands while sliding**.

- 3 In the command window, enter `variable set variable=.Latch.DV_11 real=$slider_1`.

The command sets the variable value of DV_11, the variable assigned to the down force value, to be equal to the slider selection.

- 4 Select **Apply**.

- 5 From the Dialog-Box Builder **Attributes** option menu, select **Value**.

- 6 Set the **Value** of the slider to **80**, the **Min. Value** of the slider to **60**, and the **Max. Value** to **90**.

This matches the range of values that was automatically set for DV_11 when you created it. If you want to open up the range to different values, you need to first modify the set of allowable values for the design variable.

- 7 Select **Apply**.

- 8 Repeat the above seven steps for the Up Force Value slider, replacing the command for this slider with `variable set variable=.Latch.DV_12 real=$slider_2`.

Make sure you use the numerical values you assigned when creating the design variable used for this slider (**Value** = 10, **Min. Value** = 0, **Max. Value** = 20).

- 9 Select **Apply**.

Testing the Dialog Box

Use the Test Box command to keep the Dialog-Box Builder open and operational in test mode. When the dialog box is in test mode you can issue commands from it, but you can also quickly go back to edit mode by selecting Test Box again or double-clicking on the dialog box background.

To test the dialog box:

- From the Dialog-Box Builder **Options** menu, select **Test Box**.

Saving Your Dialog Box

Save your dialog box in a command file. In ADAMS/View, a command file is simply a file that contains a set of ADAMS/View commands.

To save your dialog box:

- 1 To enter edit mode, double-click on your dialog box background.
- 2 From the Dialog-Box Builder, select **Dialog Box**, point to **Export**, and then select **Command File**.

Your dialog box is saved in a command file. You can access it through the Database Navigator.

Modifying Force Values


You now modify the clamping and release force values to find the optimal force magnitude. You also plot the force values to see at what point the handle comes down to rest on top of the hook and when the `overcenter_angle` measure drops slightly below zero, creating a toggle force.

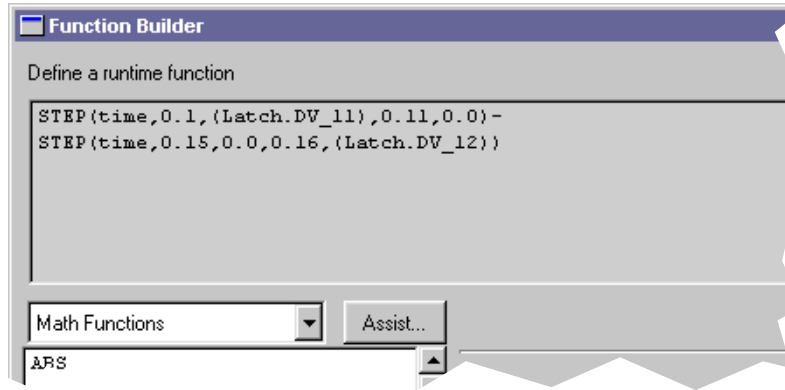
You can try different settings for the clamping and release forces to verify that the handle can be closed and released per the requirements.

Note: If the sensor is causing problems with the investigation, you can deactivate it:

- 1 Click the background to ensure that nothing is selected.
- 2 From the **Edit** menu, select **Deactivate**.
The Database Navigator appears.
- 3 From the Database Navigator, select the sensor.

To modify the clamping and release force values:

- 1 Right-click the single-component force (SFORCE) icon, point to **Force:SFORCE_1**, and then select **Modify**.
The Modify Force dialog box appears.
- 2 Next to the **Function** text box, select the **More tool** .
The Function Builder appears.
- 3 Clear the function text area located at the top of the Function Builder.
- 4 In the function text area, enter the STEP function as shown next:



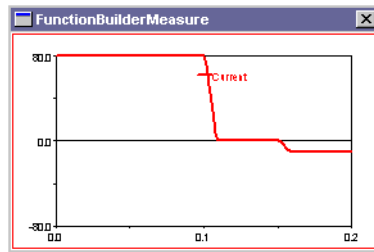
- 5 Select Apply.

Plotting the force values:

- 1 From the Function Builder, select **Plot Limits**.
The Function Builder Plot Limits dialog box appears.
- 2 Change the **End Value** to 0.2.
- 3 Select OK.

4 Select Plot.

The Function Builder Measure plot displays the force function. This is the handle force you apply to the latch. The Down Force Value slider sets the clamping force. The Up Force Value slider sets the release force.



The Next Step

You have now finished designing the latch model to meet the design requirements, without first having to build a physical prototype. For more information about using ADAMS/View, see the [ADAMS/View Online Guides](#).

Index

A - B

Angle measure

 creating 36

 markers in 37

 plot of 37

Binary files, described 27

Box tool, using 31

Box, creating 31

C - D

Clamping force

 modifying 77

 plotting 77

Command files, described 76

Contact tool, using 31

Database Navigator

 design variables in 65

 spring force in 35

 spring force measure in 66

Database, creating 14

Design Evaluation Tools

 dialog box 58

 using 57

Design process steps

 flowchart 7

 use for latch model 6

A - B

C - D

E - F

G - H

I - J

K - L

M - N

O - P

Q - R

S - T

U - V

W - Z

Design requirements 8

Design study

- plots resulting from 59
- results 61
- running 57

Design variables

- creating 50, 72
- in Table Editor 53
- limits 64
- modifying 56, 64
- modifying, dialog box 65
- reviewing values 52

Dialog-Box Builder, using 73

Down force value, modifying 77

Dynamic Pick tool, using 18

Dynamic Translate tool, using 33

E - F

Extrusion tool, using 24

Fit tool, using 32

Force

- down force value 74
- modifying, values 77
- up force value 74

Force - Single-Component tool, using 34

Function Builder Measure, plot of 79

A - B

C - D

E - F

G - H

I - J

K - L

M - N

O - P

Q - R

S - T

U - V

W - Z

G - H

Geometry

- box 31
- extrusion 24, 25, 31, 32, 33
- link 21
- plate 19
- point 18

Ground_Block, creating 31

Handle

- creating 21
- renaming 21

Handle force, creating 34

Help, getting online 16

Hook

- creating 24
- renaming 25

I - J

Icons, changing settings 17

Importing, physical test data 42

Information window, contents of 69

Joints

- inplane joint primitive, described 31
- revolute, described 21

K - L

Latch

- described 9
- design requirements 8
- illustration of physical model 8
- illustration of virtual model 10

Link tool, using 21

A - B

C - D

E - F

G - H

I - J

K - L

M - N

O - P

Q - R

S - T

U - V

W - Z

M - N

Main Toolbox

- described 15
- displaying 15

Measure

- angle, creating 36
- spring force, creating 35

O - P

Online help, about 16

Optimization

- dialog box 67
- running 66

Overcenter_angle

- creating 36
- graphical representation of 37
- markers in 37

Physical test data

- creating plot 43
- importing 42
- plot of 45

Pivot

- creating 20
- renaming 20

Plate tool, using 19

Plot Builder, using 43

Plots

- comparing 46
- layout, modifying 45

A - B

C - D

E - F

G - H

I - J

K - L

M - N

O - P

Q - R

S - T

U - V

W - Z

Plotting window
 modifying elements in 44
 using 43

Point tool, using 18

Q - R

Renaming
 handle 21
 hook 25
 pivot 20
 slider 26

Reset tool, using 22

Revolute joint
 described 21
 using 21

Revolute joint tool, using 21

S - T

Saving
 binary files 27
 command files 76
 dialog box 76
 modeling database 27

Sensor
 creating 38
 dialog box 38

Simulation tool, using 22

Simulation, described 22

Slider
 creating 26
 renaming 26

A - B

C - D

E - F

G - H

I - J

K - L

M - N

O - P

Q - R

S - T

U - V

W - Z

Spring force
 creating measure of 35
 graph of 35

Spring force measure
 creating 35
 plot of 35

Starting ADAMS
 in the UNIX environment 14
 in the Windows environment 14

Table Editor
 using, for points 51
 using, for real variables 53

Tools

- Box** 31
- Dynamic Pick** 18
- Dynamic Translate** 33
- Extrusion** 24
- Fit** 32
- Force** 34
- Inplane Joint Primitive** 31
- Link** 21
- Plate** 19
- Point** 18
- Reset** 22
- Revolute Joint** 21
- Simulation** 22
- Translational Spring-Damper** 33
- Undo** 25

A - B

C - D

E - F

G - H

I - J

K - L

M - N

O - P

Q - R

S - T

U - V

W - Z

Tools, in Main Toolbox 15

Translational Spring-Damper tool, using 33

U - V

Undo tool, using 25

Units, setting 16

Up force value, modifying 77

Virtual Test Data

 create plot of 46

 plot of 46

W - Z

Working grid

 settings 17

 zooming in on 18

A - B

C - D

E - F

G - H

I - J

K - L

M - N

O - P

Q - R

S - T

U - V

W - Z