



Adams 2022.3

Getting Started: Adams View

## Americas

5161 California Ave, Suite 200  
University Research Park  
Irvine, CA 92617  
Telephone: (714) 540-8900  
Email: [americas.contact@mscsoftware.com](mailto:americas.contact@mscsoftware.com)

## Europe, Middle East, Africa

Am Moosfeld 13  
81829 Munich, Germany  
Telephone: (49) 89 431 98 70  
Email: [europe@mscsoftware.com](mailto:europe@mscsoftware.com)

## Japan

KANDA SQUARE 16F  
2-2-1 Kanda Nishikicho, Chiyoda-ku  
Tokyo 101-0054, Japan  
Telephone: (81)(3) 6275 0870  
Email: [MSCJ.Market@mscsoftware.com](mailto:MSCJ.Market@mscsoftware.com)

## Asia-Pacific

100 Beach Road  
#16-05 Shaw Tower  
Singapore 189702  
Telephone: 65-6272-0082  
Email: [APAC.Contact@mscsoftware.com](mailto:APAC.Contact@mscsoftware.com)

## Worldwide Web

[www.mscsoftware.com](http://www.mscsoftware.com), [www.hexagon.com](http://www.hexagon.com)

## Support

<https://simcompanion.hexagon.com/>

## Disclaimer

This documentation, as well as the software described in it, is furnished under license and may be used only in accordance with the terms of such license.

Hexagon reserves the right to make changes in specifications and other information contained in this document without prior notice.

The concepts, methods, and examples presented in this text are for illustrative and educational purposes only, and are not intended to be exhaustive or to apply to any particular engineering problem or design. Hexagon assumes no liability or responsibility to any person or company for direct or indirect damages resulting from the use of any information contained herein.

User Documentation: Copyright © 2022 Hexagon AB and/or its subsidiaries. All Rights Reserved.

This notice shall be marked on any reproduction of this documentation, in whole or in part. Any reproduction or distribution of this document, in whole or in part, without the prior written consent of Hexagon is prohibited.

This software may contain certain third-party software that is protected by copyright and licensed from third party licensors. Additional terms and conditions and/or notices may apply for certain third party software. Such additional third party software terms and conditions and/or notices may be set forth in documentation and/or at <http://www.mscsoftware.com/thirdpartysoftware> (or successor website designated by Hexagon from time to time). Portions of this software are owned by Siemens Product Lifecycle Management, Inc. © Copyright 2022.

The Hexagon logo, Hexagon, MSC, MSC Adams, MD Adams, and Adams are trademarks or registered trademarks of Hexagon AB and/or its subsidiaries in the United States and/or other countries. FLEXIm and FlexNet Publisher are trademarks or registered trademarks of Flexera Software. Parasolid is a registered trademark of Siemens Product Lifecycle Management, Inc. All other trademarks are the property of their respective owners.

ADAM:V2022.3:Z:Z:DC-GS-VIEW

# Documentation Feedback

At Hexagon Manufacturing Intelligence, we strive to produce the highest quality documentation and welcome your feedback. If you have comments or suggestions about our documentation, write to us at: [documentation-feedback@mscsoftware.com](mailto:documentation-feedback@mscsoftware.com).

Please include the following information with your feedback:

- Document name
- Release/Version number
- Chapter/Section name
- Topic title (for Online Help)
- Brief description of the content (for example, incomplete/incorrect information, grammatical errors, information that requires clarification or more details and so on).
- Your suggestions for correcting/improving documentation

You may also provide your feedback about Hexagon Manufacturing Intelligence documentation by taking a short 5-minute survey at: <http://msc-documentation.questionpro.com>.

**Note:**

The above mentioned e-mail address is only for providing documentation specific feedback. If you have any technical problems, issues, or queries, please contact [Technical Support](#).



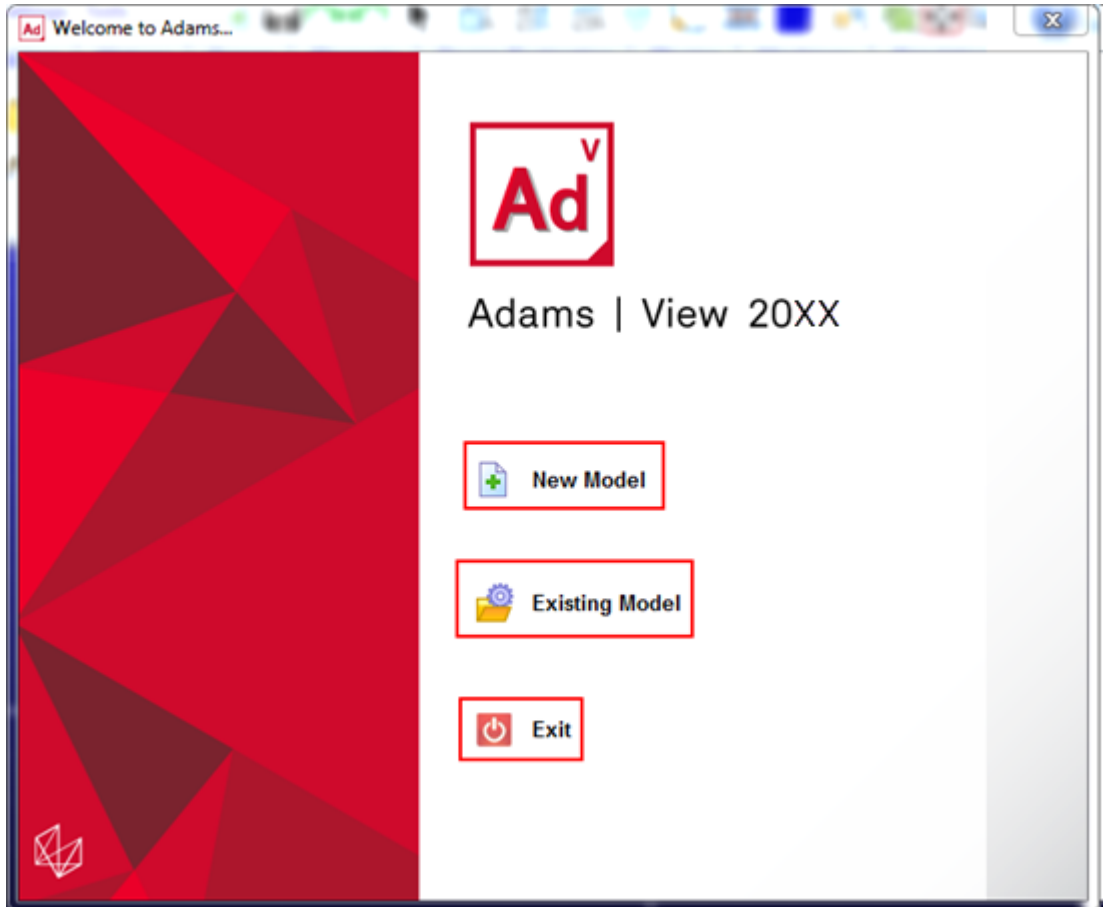
# Getting Started Using Adams View

# Welcome Screen


## Welcome to Adams View Start Screen

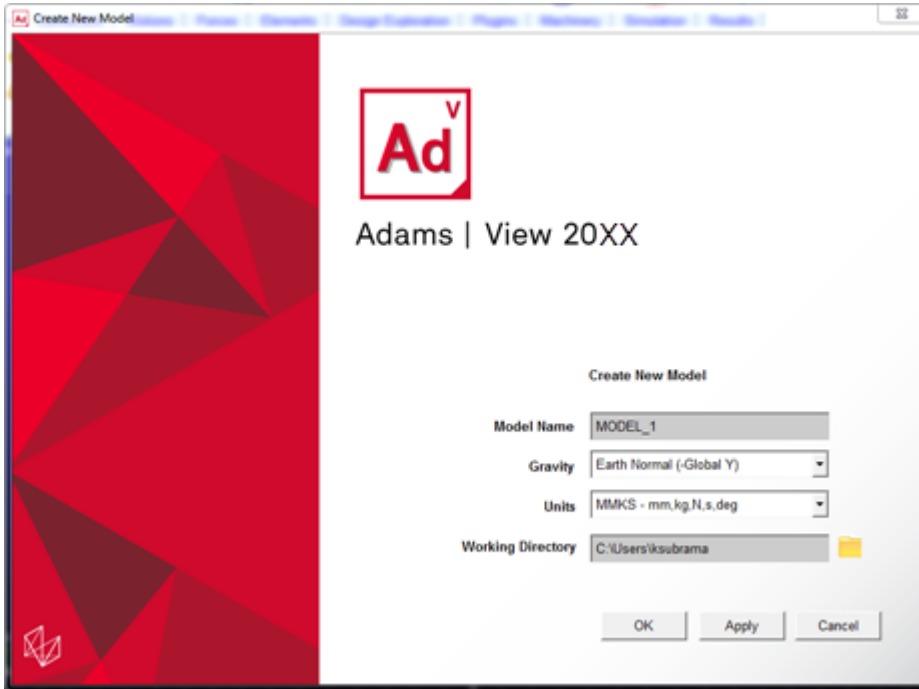
Adams View is a powerful modeling and simulating environment. You can use Adams View to build and simulate a model of any mechanical system that has moving parts. You can test your model in the same environments your actual product will experience. You can also validate your model against test data, 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.

With the Adams 2012 release, the updated Graphical User Interface (GUI) introduced. This includes changes to the Start Screen where you are provided with the option to create new model or import an existing model.




## New Model


To create a new model, select the **New Model** icon  from the start screen. You will be presented a panel which consist of options needed to create a new model.



You can change the working directory by clicking the **Browse For Folder**  button.

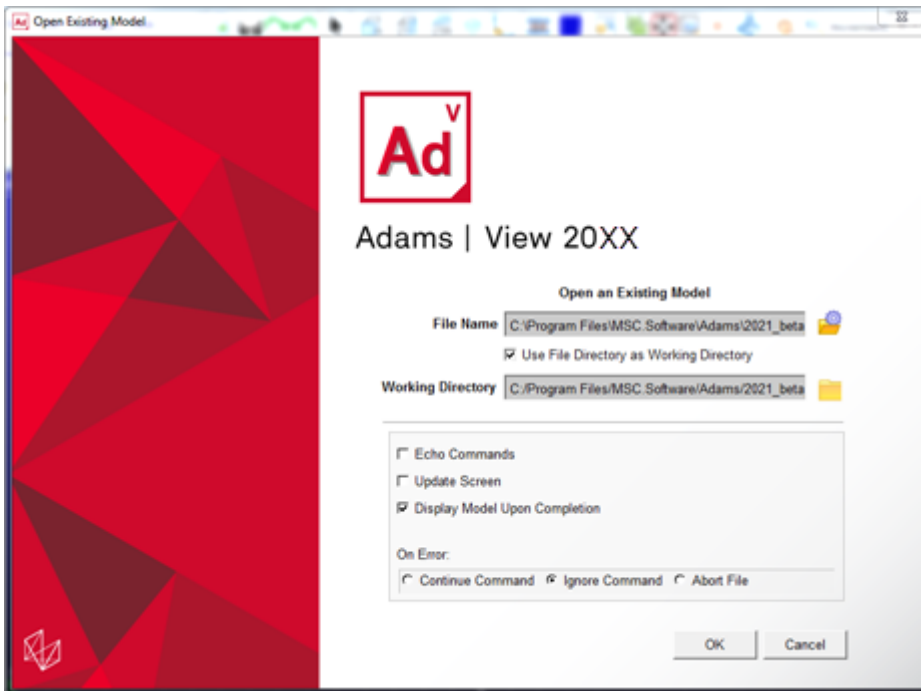
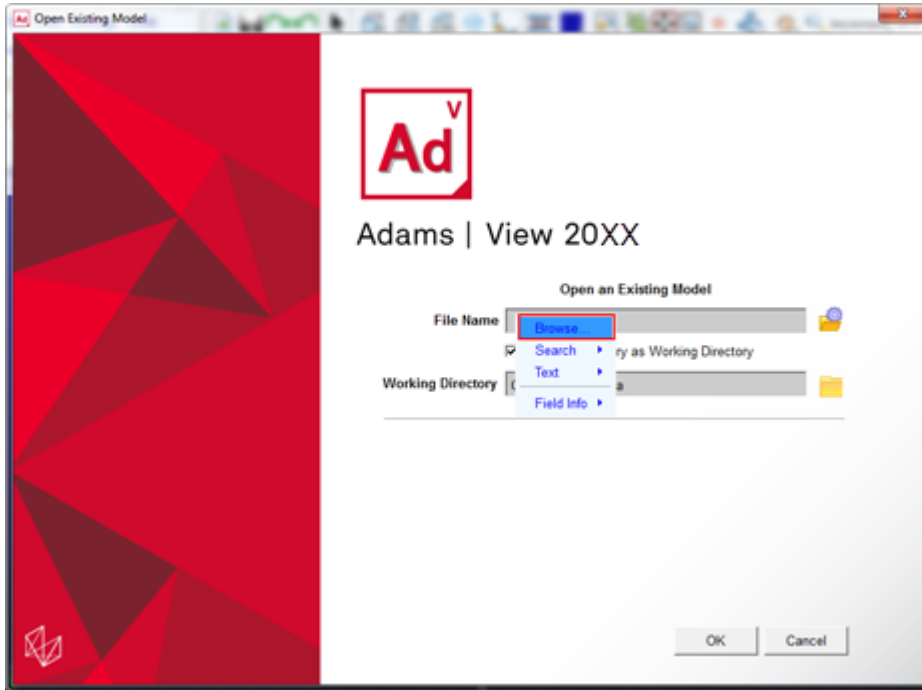
## Existing Model

To start your Adams session from an existing model, select the **Existing Model** icon  from the start screen.

You can browse for the model by clicking the **Existing Model**  button in the opened dialog box or you can right click inside the **File Name** field and locate the existing model through the **Browse** option.

## 4 Getting Started Using Adams View

Welcome to Adams View Start Screen





## 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, 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 additional modules, such as Adams Flex, and Adams Controls.



# 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.

**Note:** By default on Windows, files in the installation directory are read-only. During installation, your system administration can choose to change the permissions so you can write to the installation directory. If this has not been done, you need to change the permissions of the files when you copy them to your working directory.



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

This chapter contains the following sections:

- [What You Will Learn](#)
- [What You Will Create](#)

## What You Will Learn

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

### Build a model by:

- Including moving parts, joints, flexible connections, and applied forces.

### Test your design by:

- Performing simulations as you build up your model.
- Simulating the motion of the model as it would perform in actual operation.

### Review your test by:

- Viewing the results of the simulations.
- Comparing physical test data with the results of the virtual simulation.

**Improve your model by:**

- Refining your model so your virtual test data matches your physical test data.
- Iterating your design to assess the sensitivity of its performance to various design changes.
- Optimizing your design to find the best combination of design parameters to achieve maximum performance.

Finally, you can automate the design process steps so you can quickly evaluate various design options.

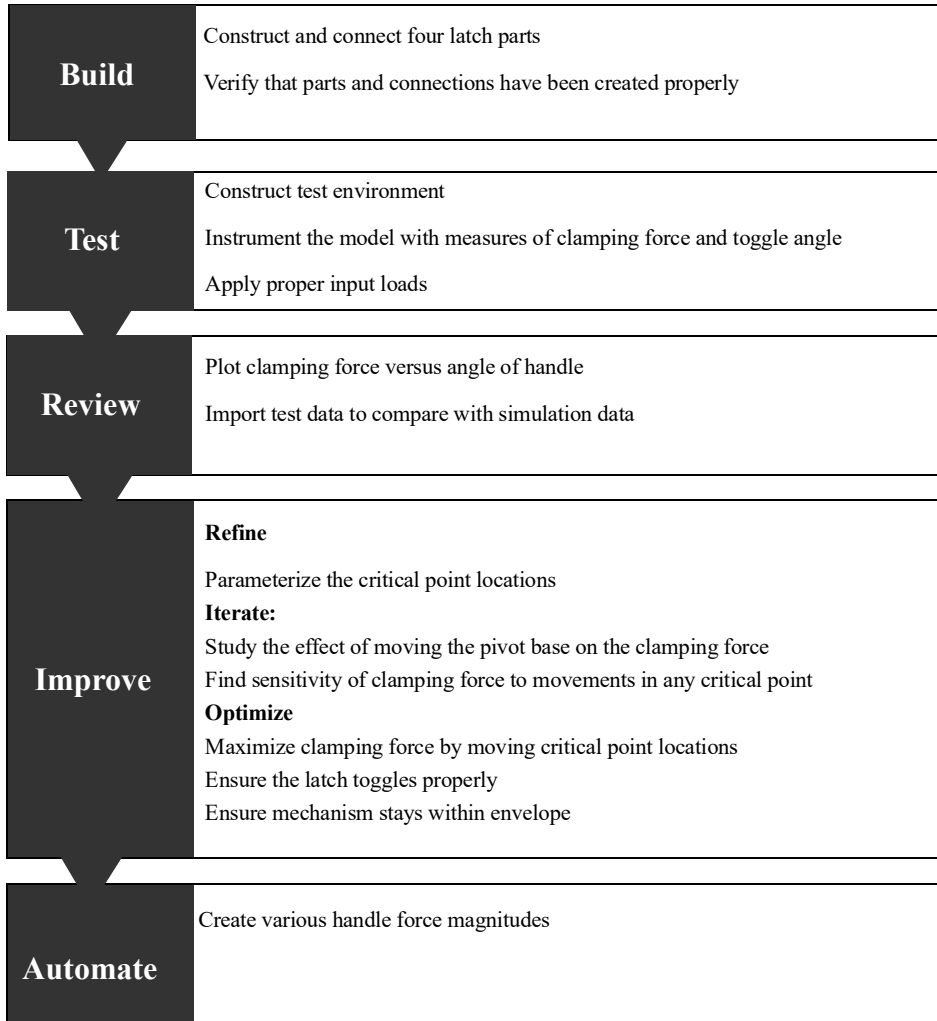


Figure 1 Design Process Steps for the Latch Model

## What You Will Create

In this tutorial, you will 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.

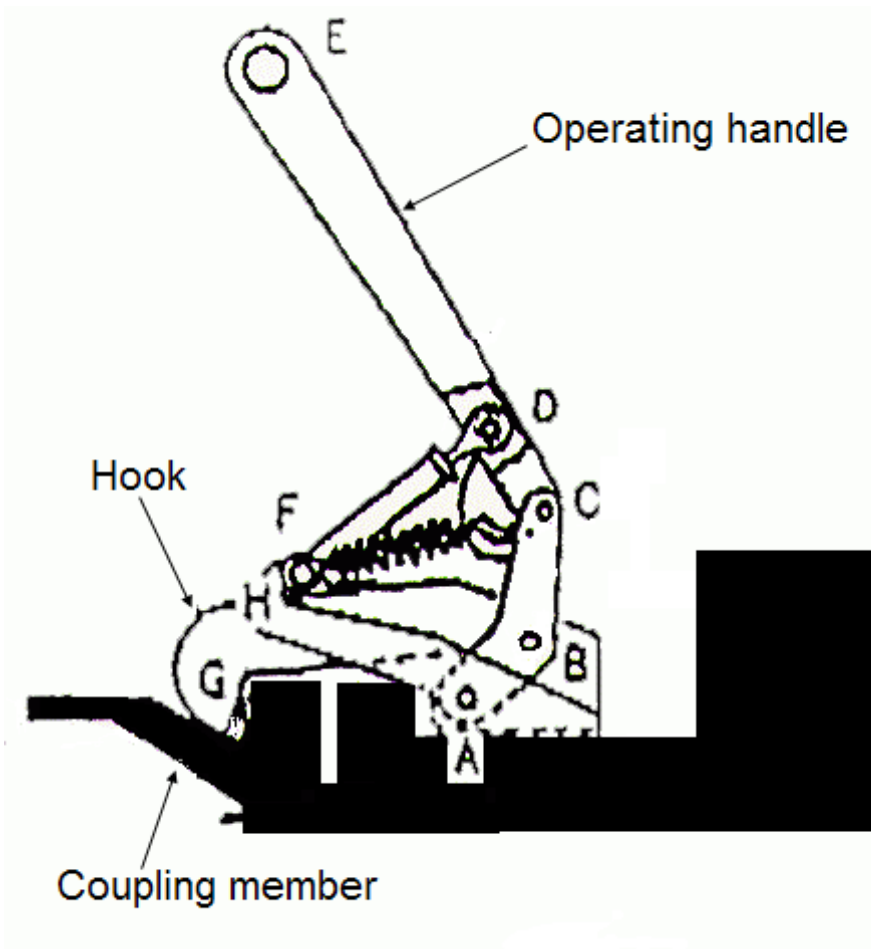


Figure 2 Physical Model of Hand Latch Design

## 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 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.

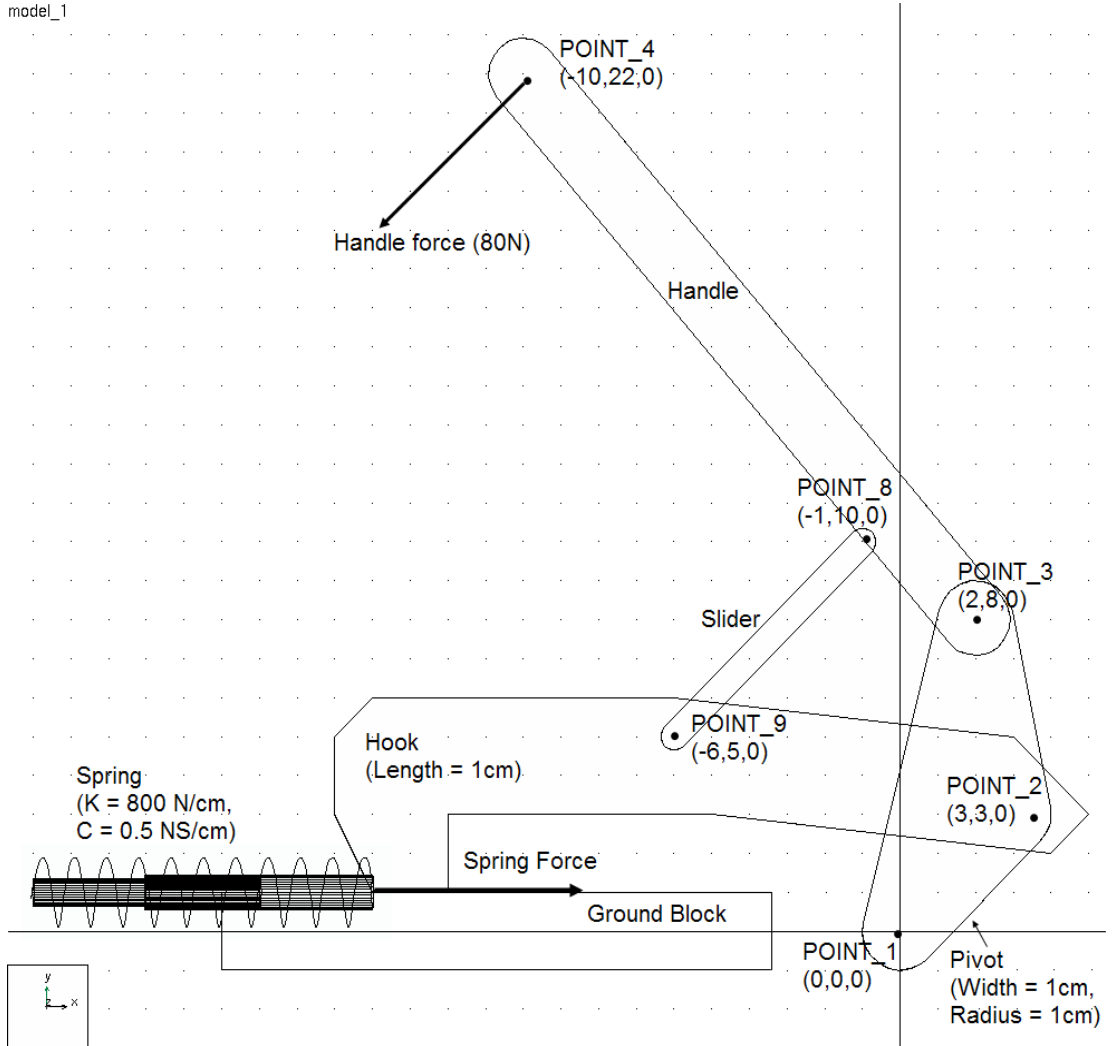


Figure 3 Adams View Latch Model

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 need to change the relative locations of the points to see their effect on the design requirements.





# Building Your Model

## Overview

In this chapter you build and connect the parts of the toggle latch. You also verify that, you have created 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](#).

Adams View assigns to the parts you create the material type of steel, with a material density of  $7801.0 \text{ kg/m}^3$ .

You build the latch model in two basic sections:

- Building the Pivot and Handle
- Building the Hook and Slider

Figure 1 shows the latch as it should look when you have finished it.

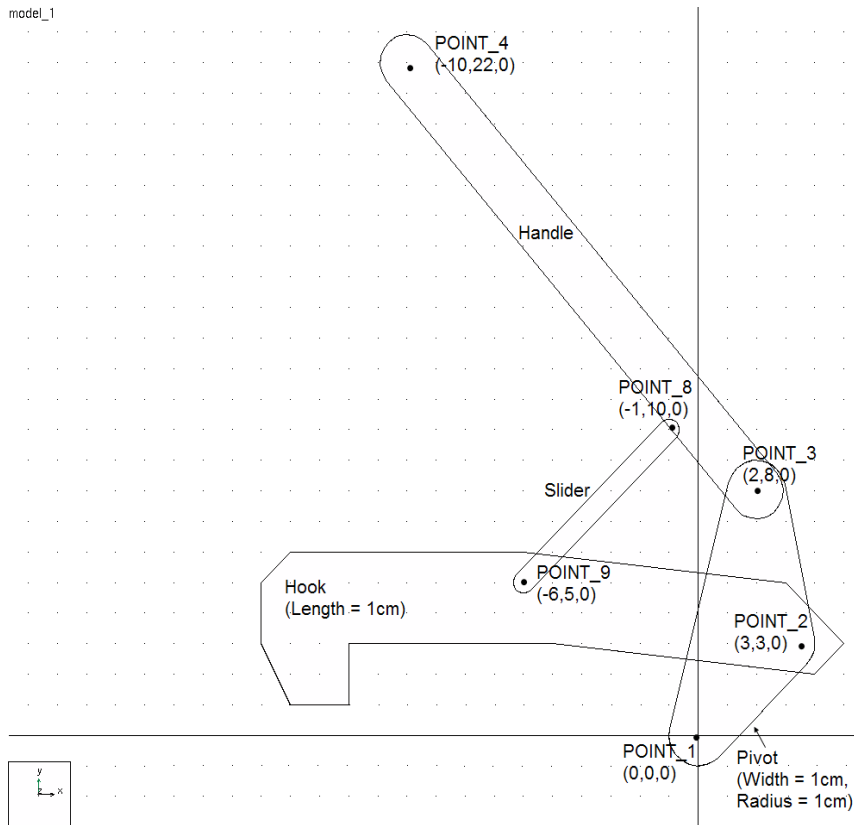


Figure 1 Latch in Build Phase

## 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](#)
- [Familiarizing Yourself with Adams View](#)
- [Setting Up Your Work Environment](#)
- [Creating Design Points](#)
- [Creating the Pivot](#)
- [Renaming the Pivot](#)
- [Creating the Handle](#)
- [Connecting the Parts Using Revolute Joints](#)
- [Simulating the Motion of Your Model](#)
- [Seeing the Effect of Parameterization](#)

### 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 Linux environment you start Adams View from the Adams Toolbar. In the Windows environment you start Adams View from the Start button.

#### To start Adams View in the Linux environment:

1. At the command prompt, enter the command to start the Adams Toolbar, and then press **Enter**. The standard command that MSC Software provides is `adamsx`, where `x` is the version number, for example `adams2022_3`.

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 2022.3**, 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, ribbon based tabbed icon containers, see section [Ribbon Capability](#) for more information.

**Ribbon: Tabbed icon containers**

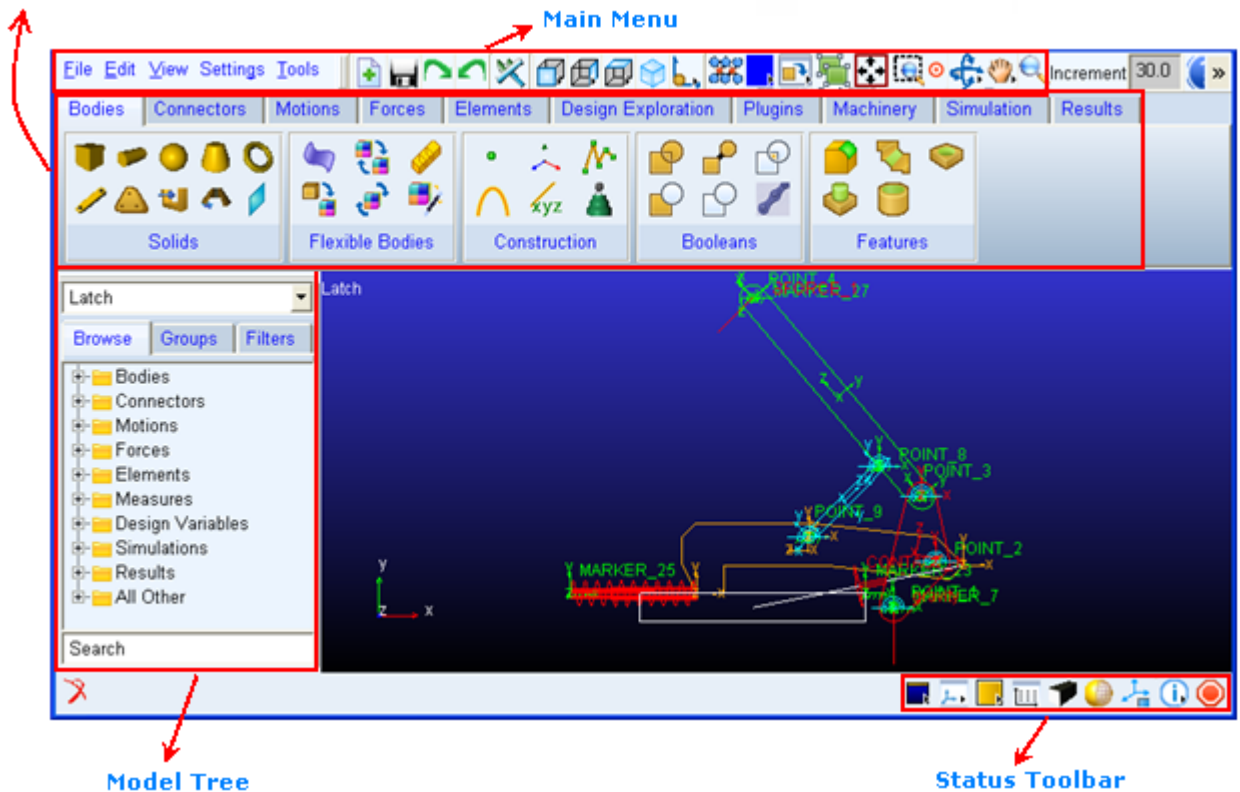


Figure 2 Adams View interface

## Learning About Getting Help

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

- Use the online help to read overviews, procedures, and see many examples. From the **Help** menu, select **Adams View Help**.

- Use the dialog box 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.

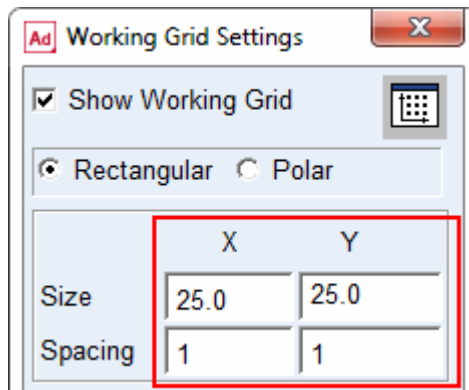
For more information about getting help in Adams View, see the printed *Release Guide* that comes with your license.

## 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 for X and Y to 1



**Note:**

You do not need to enter units because Adams View uses the default units if you do not specify them. 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. Near the bottom of the Icon Settings dialog box set **Name Visibility** to **On**.
10. 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 in the Coordinates window and next to the cursor.

During this tutorial, you will find it very helpful to have the Coordinates window open because you will place many objects based on their coordinate location.

## 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 menu, select the **Dynamic Pick** tool  to zoom in on your grid.
2. Drag the cursor across the area of the working grid display.
3. Click the **Bodies** tab on the Adams View ribbon.
4. From the **Construction** container, click the **Point** tool  and click the locations shown in [Table 1](#) 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 1](#).

### To create the pivot:

1. Click the **Bodies** tab on the Adams View ribbon.
2. From the **Solids** container, click the **Plate** tool .
3. In the Main toolbox, 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.

4. In the **Radius** text box, enter **1**, and then press **Enter**.
5. 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 1](#).

6. Right-click to stop selecting points and create the pivot geometry.

## Renaming the Pivot

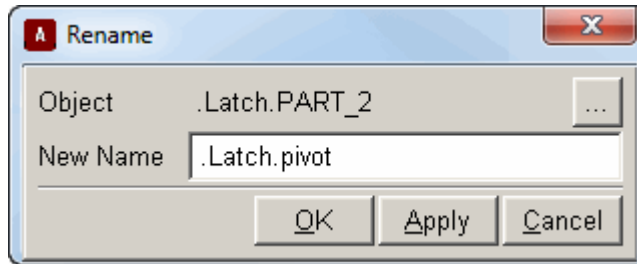
As you create objects, Adams View automatically assigns names to them. Each name consists of a string containing the object type and a unique integer ID for that type. For example, Adams View named the recently created part **PART\_2**.

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 shortcut 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. Click the **Bodies** tab on the Adams View ribbon.
2. From the **Solids** container, select the **Link** tool .
3. 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.

4. Rename the link part, **Part: PART\_3**, to **handle**, to represent the handle part as shown in [Figure 1](#).


## 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.

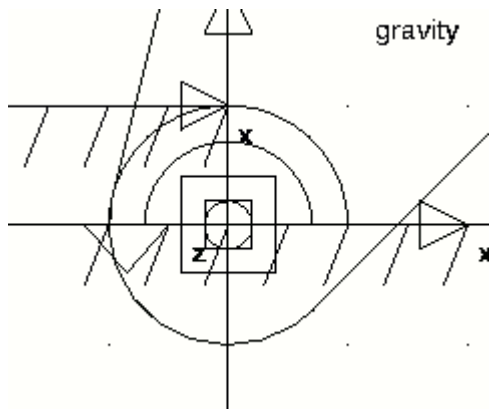
In this section, you place a revolute joint between the pivot and ground, which will allow the pivot to rotate with respect to ground. You also place a second revolute joint between the pivot and handle to allow the two parts to rotate with respect to each other.

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. Click the **Connectors** tab on the Adams View ribbon.
2. From the **Joints** container, select the **Revolute Joint** tool .
3. To select the parts to attach, click the **pivot** and **ground** (the background).
4. 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. in [Setting Up Your Work Environment](#), 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.

5. Select the **Revolute Joint** tool again.
6. 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 have 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. Click the **Simulation** tab on the Adams View ribbon.



2. From the **Simulate** container, click the **Simulate** tool  .
3. Set up a simulation with an end time of 1 second and 50 steps.
4. Select the **Simulation Start** tool  .

The model simulates, and then remains in simulate mode.

5. To return to the initial model configuration, select the **Reset** tool  .

At this step in the creation of the latch, the model is effectively a double pendulum falling due to the influence of gravity. This illustrates one of the advantages of interactive model building. Namely, that you can interrogate the behavior of the model even though the model is not complete.

## 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 `POINT_1`.  
A shortcut 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. Near the top right corner of the Table Editor, select **Apply**.  
`POINT_1`, along with the joint and the pivot, move to the new location.

<b>Note:</b>	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 the geometry of the latch model are:


- [Creating the Hook and the Slider](#)
- [Connecting the Parts Using Revolute Joints](#)
- [Simulating Your Model](#)

■ [Saving Your Database](#)

## 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. Click the **Bodies** tab on the Adams View ribbon.
2. From the **Solids** container, click the **Extrusion** tool .
3. Be sure that **Create profile by:** is set to **Points** and **Closed** is selected.
4. Select **Path:** to **About Center**.
5. In the **Length** text box, enter **1**, and then press **Enter**.
6. Click the locations listed in [Table 2](#).
7. 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.

8. Right-click to stop selecting locations and create the geometry.
9. Rename the extrusion part to **hook**.

### To create the slider:

1. Create two more design points, POINT\_8 and POINT\_8\_2, at the locations shown in [Table 3](#).

Table 3 Points Coordinate Locations

	X location:	Y location:	Z location:
POINT_8	-1	10	0
POINT_8_2	-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\_8\_2.
  - 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 shortcut 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 have 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 .2 second, 200-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**.

# 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 will be doing the following:

- [Creating the Ground Block](#)
- [Adding a Three-Dimensional Contact](#)
- [Adding a Spring](#)
- [Creating a Handle Force](#)
- [Creating a Measure on the Spring Force](#)
- [Creating an Angle Measure](#)
- [Creating a Sensor](#)
- [Saving Your Model](#)
- [Simulating Your Model](#)



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 1](#) shows how your model should look at the end of the test phase.

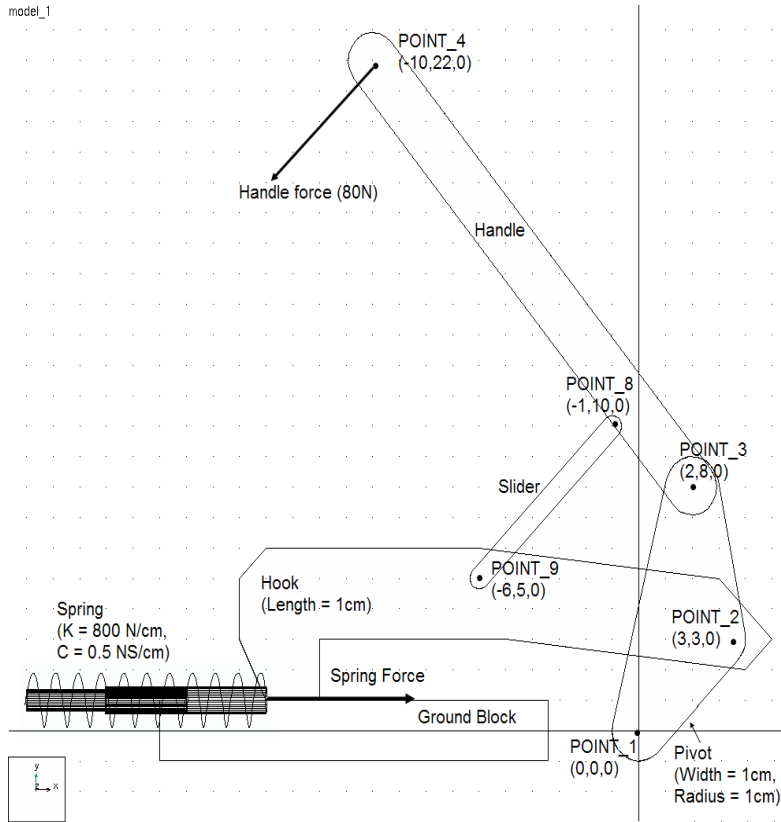



Figure 1 Latch Model in Test Phase

## 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. Click the **Bodies** tab on the Adams View ribbon.
2. From the **Solids** container, select the **Box** tool , and change its construction method from **New Part** to **On Ground**.
3. 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.
4. 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.

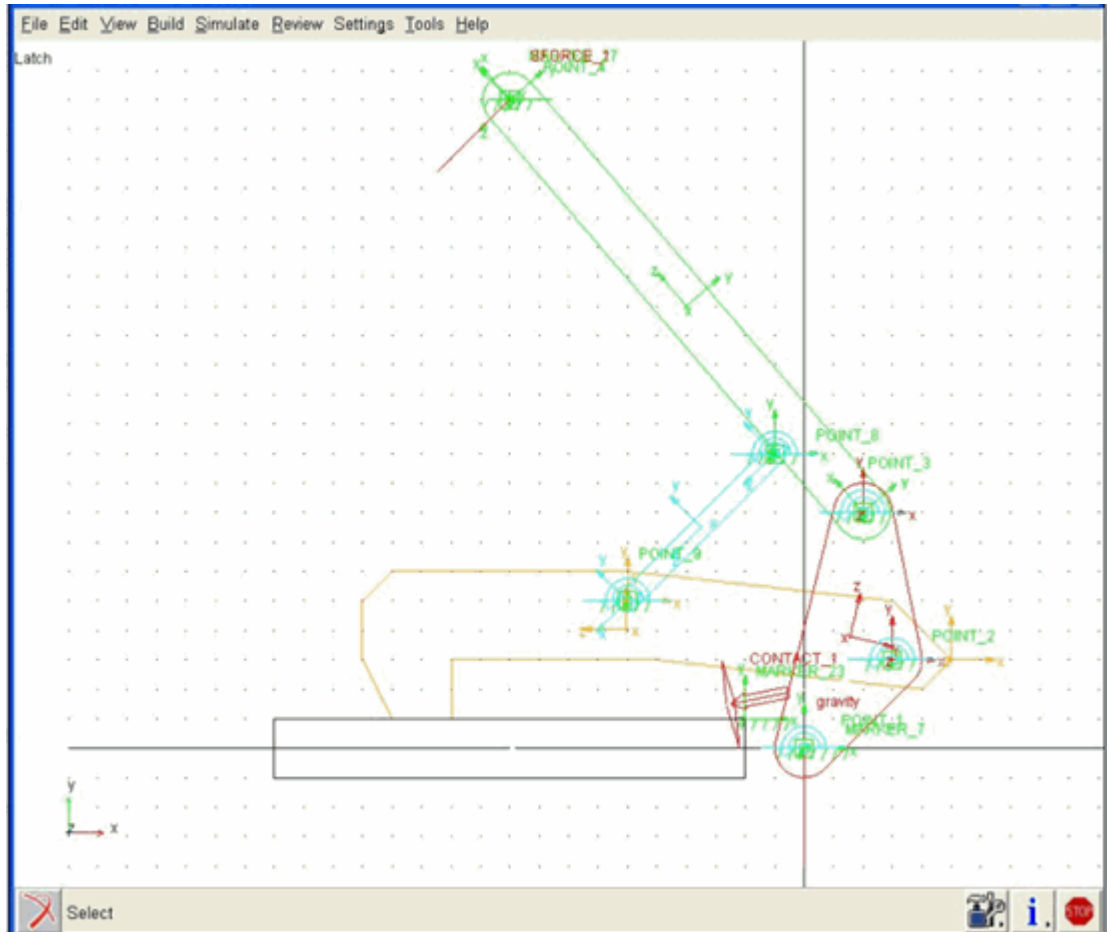
### To add a three-dimensional contact:

1. Click the **Forces** tab on the Adams View ribbon.
2. From **Special Forces** container, click the **Contact** tool .

The Create Contact dialog box appears.

3. Right-click the **I Solid** text box, point to **Contact\_Solid**, and then select **Pick**. Select the **hook** (`EXTRUSION_7`).
4. Now do the same for the **J Solid** text box, selecting the **ground\_block** (`BOX_10`).
5. Because you will use the default values for the contact force, select **OK**.
6. From the main menu, select the **Fit** tool .

Adams View fits your entire model into the current window. It should look similar to the one shown below:



## Adding a Spring



The spring between the hook and ground represents the clamping force when the hook engages the container.

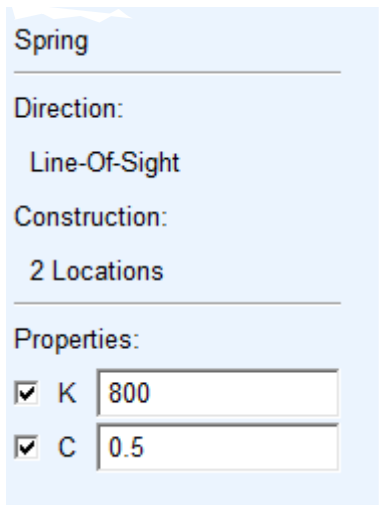
The spring should have a stiffness coefficient of  $800 \frac{\text{N}}{\text{cm}}$  and a damping coefficient of  $0.5 \frac{\text{N} \cdot \text{s}}{\text{cm}}$ , so that a hook movement of 1.0 cm gives a large clamping force.

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. From the main menu, click the **Dynamic Translate** tool , to move your model to the right, to give yourself room to add the spring.
2. Click the **Forces** tab on the Adams View ribbon.
3. From the **Flexible Connections** container, click the **Translational Spring-Damper** tool  to create a spring between the ground and the hook.
4. Set **K** to **800** and **C** to **0.5**.



5. To add the spring, click at the following locations:
  - Hook vertex, `.HOOK.EXTRUSION_7.V16`, near location  $(-14, 1, 0)$ , making sure to select the vertex and not a point near it.

**Tip:** To ensure that you select the hook vertex, right-click with the cursor near the vertex location  $(-14,1,0)$ . A select list appears with all the vertices in the area. Select vertex 16.


- $(-23, 1, 0)$

A red spring appears.

### To modify a marker:

1. Right-click the location  $(-14, 1, 0)$ .  
A shortcut menu appears.
2. Point to **Marker: MARKER\_24**, and then select **Modify**.  
The Marker Modify dialog box appears.


**Note:** The Marker name could be `MARKER_26`, if you have worked the tutorial from the beginning rather than starting from a `cmd` file.

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. Click the **Forces** tab on the Adams View ribbon.
2. From the **Applied Forces** container, click the **Force (Single-Component)** tool .
  - Set **Run-time Direction** to **Space Fixed**.
  - Set the **Characteristic** to **Constant**.
  - Select **Force**, and then set it to **80**.
3. Select the following in the order listed:
  - The handle
  - A marker near `POINT_4`, 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.

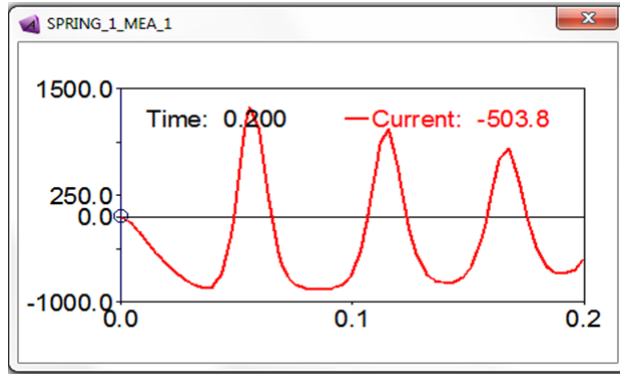
For the latch model, you need a measurement of the clamping force when the latch is clamped so you can compare it with the design requirement. The spring force measure represents the clamping force.

### 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\_8\_2. 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. Click the **Design Exploration** tab on the Adams View ribbon.
2. From the **Measures** container, click **Angle Measure** tool . Click **Advanced** button.

The Angle Measure dialog box appears.

3. In the **Measure Name** text box, enter the measure name as **overcenter\_angle**.
4. Right-click the **First Marker** text box, point to **Marker**, and then select **Pick**.

- Pick the markers to enter in your measure as shown in row 1 of [Table 1](#) and illustrated in [Figure 2](#).

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

- Repeat the above two steps for the **Middle Marker** and **Last Marker**

Table 1 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_8_2	-6, 5, 0

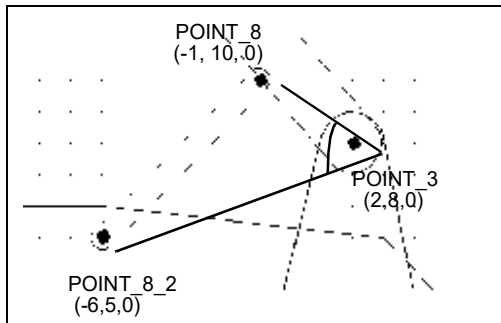
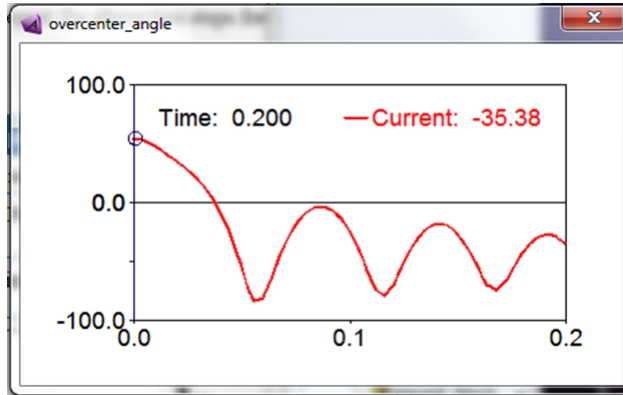


Figure 2 Graphical Representation of overcenter\_angle


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

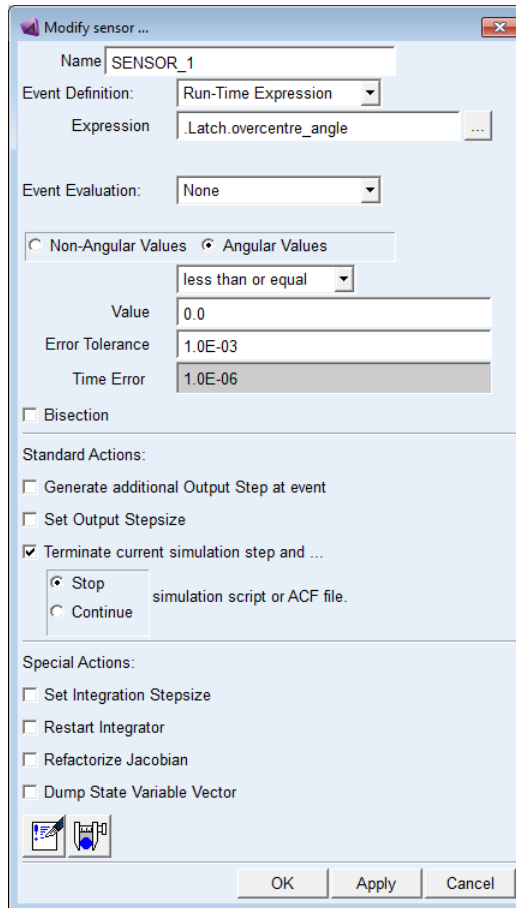


## Creating a Sensor

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. Click the **Design Exploration** tab on the Adams View ribbon.
2. From the **Instrumentation** container, click **Sensor**  icon.  
 The Create Sensor dialog box appears.
3. 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.

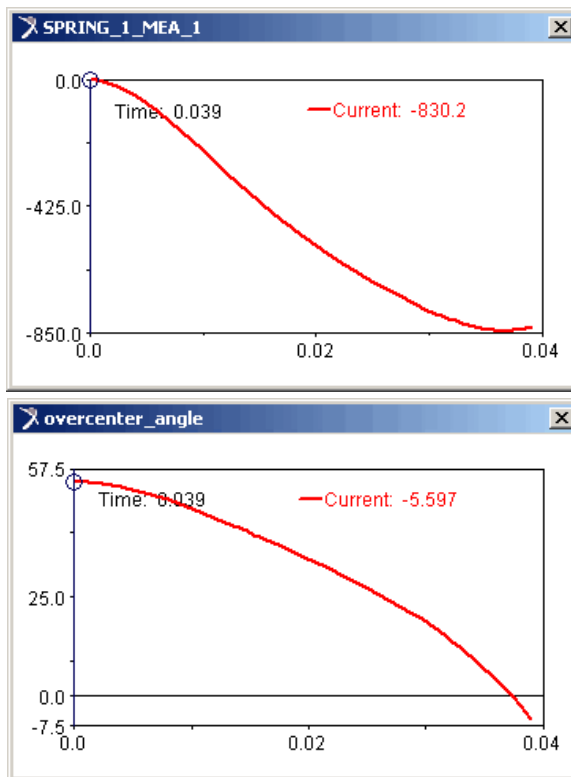
### To simulate your model:

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.





# Reviewing 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](#)
- [Creating a Plot Using Physical Test Data](#)
- [Modifying Your Plot Layout](#)
- [Creating a Plot Using Virtual Test Data](#)
- [Saving Your Model](#)



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 in [Simulating Your Model](#).

## 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. Right-click the **File to Read** text box, point to **Search**, and then select `$aview_dir/examples/Latch`.  
From the File Selection dialog box, select `test_dat.csv`.
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. Click the **Results** tab on the Adams View ribbon.
2. From the Postprocessor container, click **Postprocessor** tool  or press F8.

Adams View launches Adams PostProcessor (see [Figure 1](#)), a post-processing tool that lets you view the results of simulations you performed. Adams PostProcessor has several modes, including plotting (default), animation, and report, as shown in the first pull-down menu on the menu toolbar.

Take a minute to familiarize yourself with Adams PostProcessor. For more information, see the Adams PostProcessor online help.

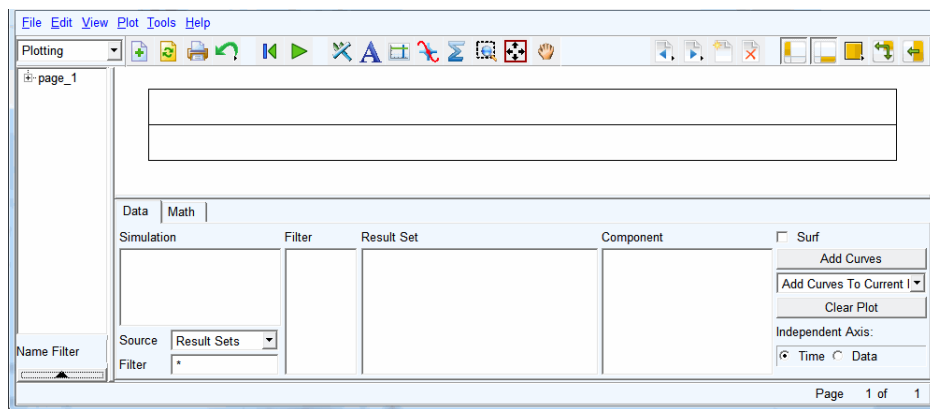
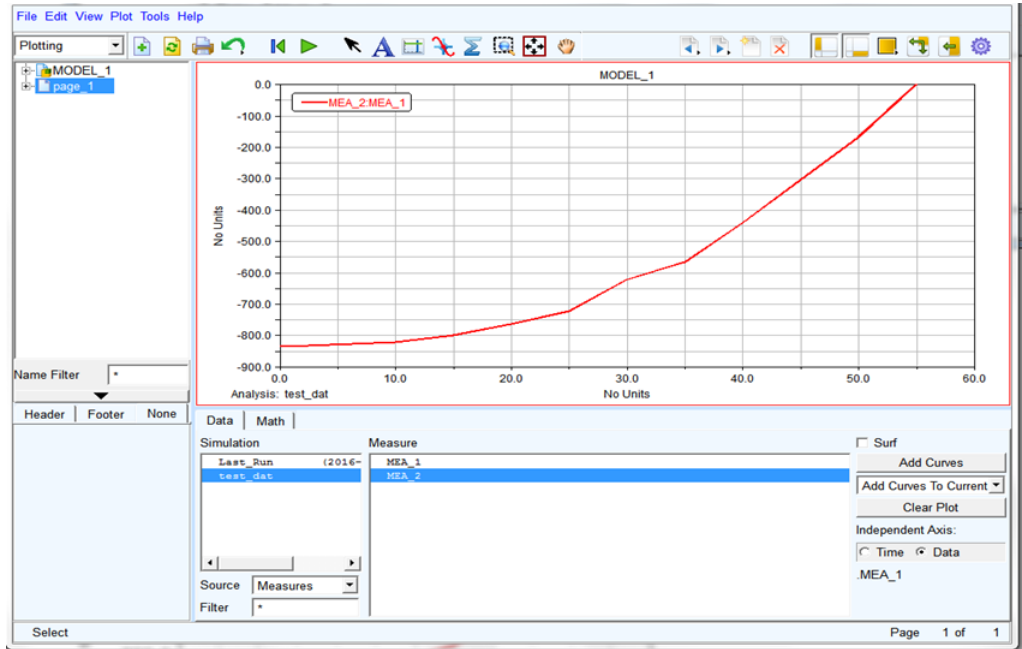


Figure 1 Adams PostProcessor

3. At the bottom left of the dashboard, set **Source** to **Measures**.
4. From the **Simulation** list, select **test\_dat**.
5. 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.
6. Select **MEA\_1**.
7. Select **OK**.
8. From the dashboard, from the **Measure** list, select **MEA\_2**, for the vertical axis data.
9. Select **Add Curves** to add the new data to the plot.  
 Adams PostProcessor displays the plot of x against y as shown next:



## Modifying Your Plot Layout

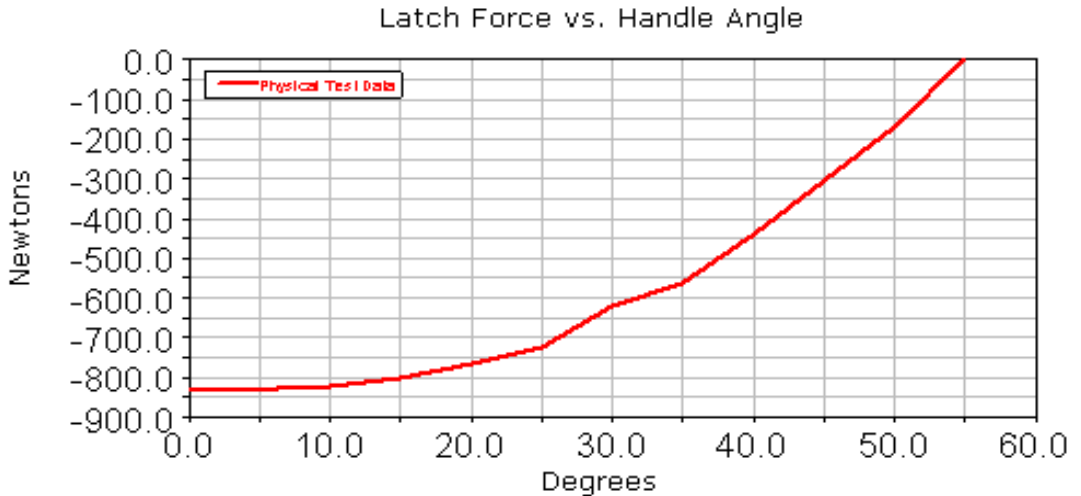
To make your plot ready for a report, you assign a title to the plot, assign units to the x- and y-axis, and modify the plot legend.

### To assign units:

1. From the treeview, double-click **plot\_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**, and then press **Enter**.  
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.

### To create a plot using virtual test data:

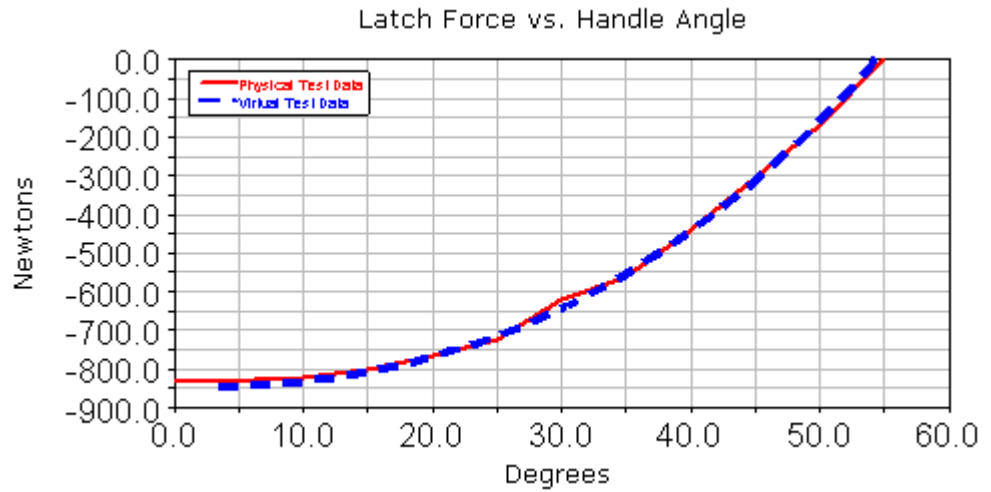
1. In the dashboard, from the **Simulation** list, select **Last\_Run (...)**.
2. Set **Independent Axis** to **Data**.

The Independent Axis Browser appears.

**Note:** In case the Independent Axis browser does not appear, set the Independent Data to "**Time**" and then back to "**Data**".

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**.

- Change the legend text for this curve (curve\_2) to **Virtual Test Data**.



- 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**.



# Improving 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](#)
- [Reviewing Design Variable Values](#)



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

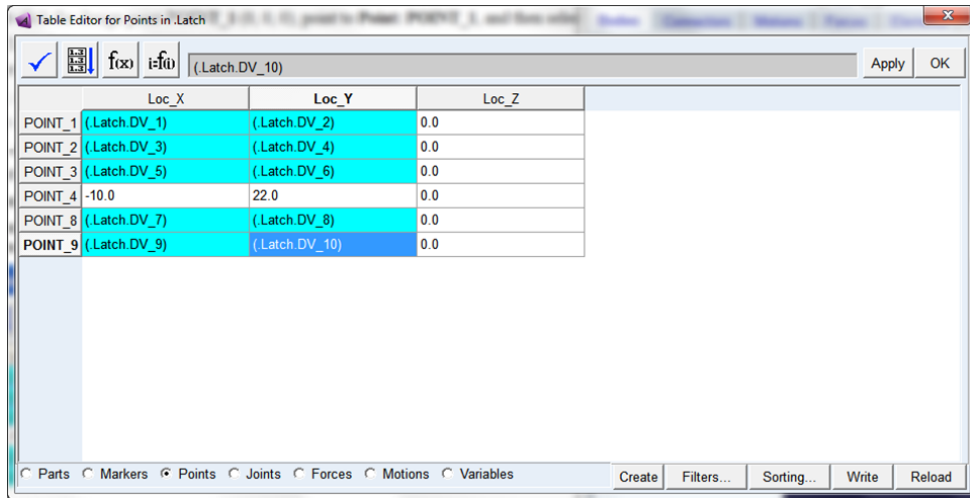
You now create design variables to represent the design points in your model, except for `POINT_4`, the handle end point. You do not modify `POINT_4`, because doing so would not meet envelope requirements for the mechanism. In subsequent chapters you will use design variables to run design studies and 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. To set the display of the Table Editor, click any cell other than the top left cell.
3. Now, double-click the **Loc\_X** cell for `POINT_1`.
4. 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.

5. Select the **Loc\_Y** cell for `POINT_1`.
6. Repeat Step 4.
7. Repeat the above procedure for the x and y locations of `POINT_2`, `POINT_3`, `POINT_8`, and `POINT_8_2`. Your Table Editor dialog box should look like the one shown here:



**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.

8. 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 have 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 [Iterating Your Design](#)). 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
DV_1	0.0	-1.0, 1.0	yes	absolute
Latch.SPRING_1.stiffness_coefficient	800.0	(NONE), (NONE)	yes	absolute
Latch.SPRING_1.damping_coefficient	0.5	(NONE), (NONE)	yes	absolute
Latch.SPRING_1.free_length	1.0	(NONE), (NONE)	yes	absolute
Latch.SPRING_1.preload	0.0	(NONE), (NONE)	yes	absolute
DV_2	0.0	-1.0, 1.0	yes	absolute
DV_3	3.0	-10.0, 10.0	yes	percent_relative
DV_4	3.0	-10.0, 10.0	yes	percent_relative
DV_5	2.0	-10.0, 10.0	yes	percent_relative
DV_6	8.0	-10.0, 10.0	yes	percent_relative
DV_7	-1.0	-10.0, 10.0	yes	percent_relative
DV_8	10.0	-10.0, 10.0	yes	percent_relative
DV_9	-6.0	-10.0, 10.0	yes	percent_relative
DV_10	5.0	-10.0, 10.0	yes	percent_relative

6. Select **OK**.
7. Save your the modeling database as **refine**.



# 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](#)
- [Running a Design Study](#)
- [Examining the Results of Design Studies](#)




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. Click the **Design Exploration** tab on the Adams View ribbon.

2. From the **Measures** container, click **Display a Measure tool** .

The Database Navigator appears.

3. Select `SPRING_1_MEA_1`.
4. Select **OK**.
5. Run a .2 second, 100-step simulation and then return to the initial model configuration.

Adams View updates the spring measure strip chart.

6. Right-click the spring force curve in the strip chart. Point to **Curve: Current**, and then select **Save Curve**.

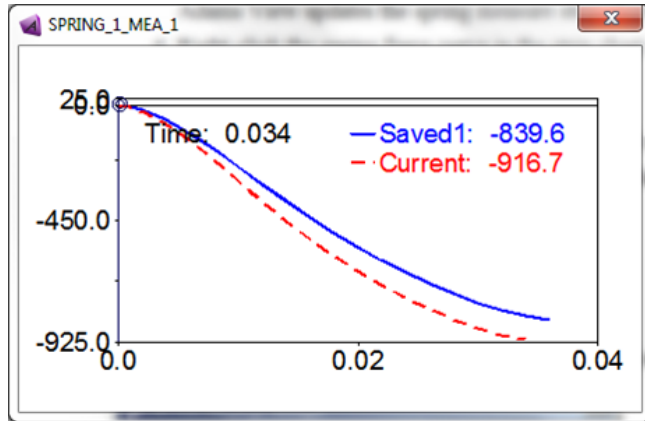
### To modify a design variable:

1. From the **Model Browser** window, click **Design Variables** and expand.
2. Select `DV_1` from the list, right click and select **Modify**.

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

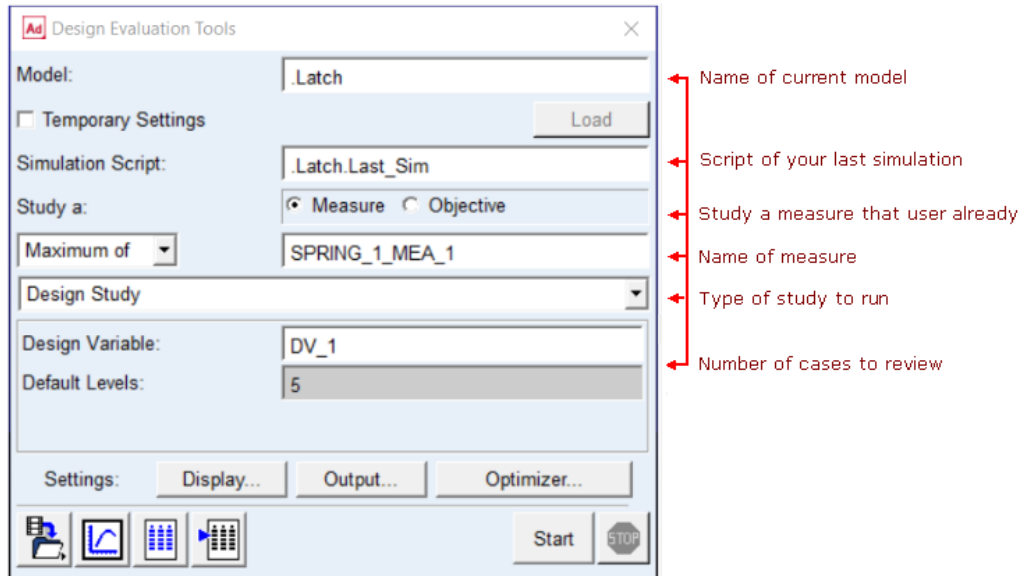
Run a design study to quickly look at a range of design variable values, and see how they affect the design. Adams View gives you the option of displaying various plots, as well as a design study report. The design study report includes the design sensitivity of the pivot location, which is the rate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

### To run a design study:

1. Click the **Design Exploration** tab on the Adams View ribbon.
2. From the **Design Evaluation** container, click **Design Evaluation tool**  .

The Design Evaluation Tools dialog box appears.

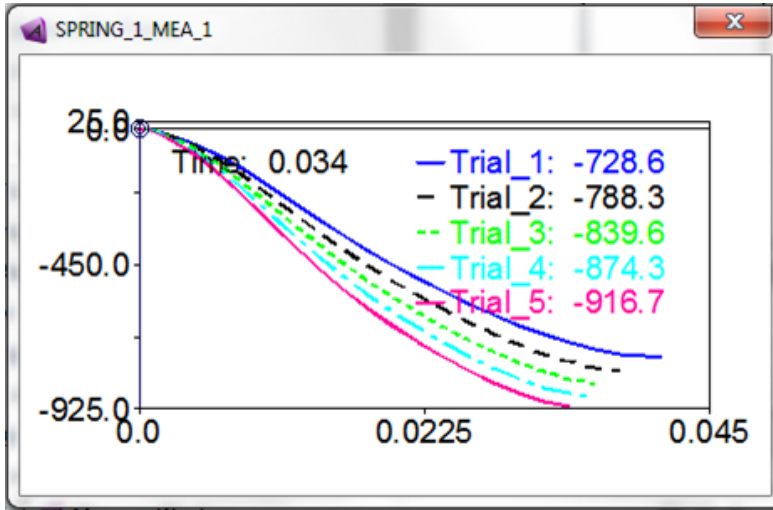
3. Fill out the dialog box that appears so it matches the one shown here. Leave the dialog box open.



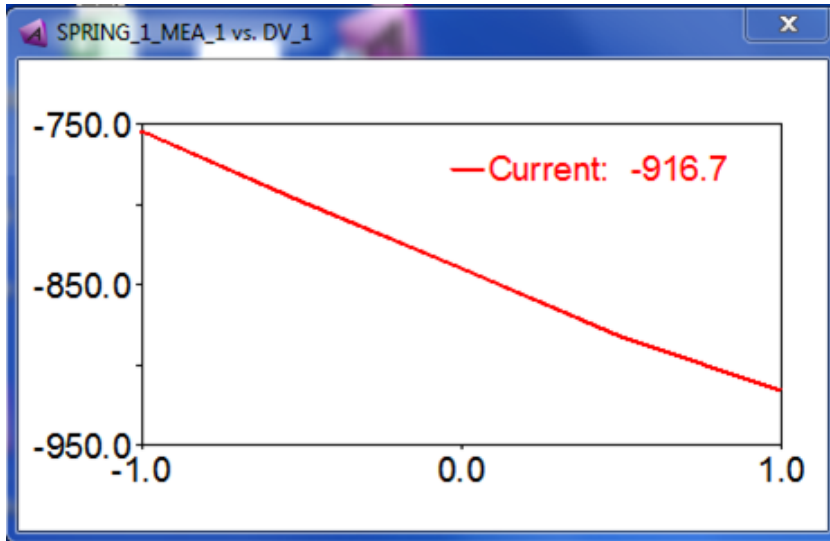
4. In the Design Evaluation Tools dialog box, select **Display**.  
The Solver Settings dialog box appears.
5. To see all the options, select **More**.
6. Set **Time Delay** to **0.0**.
7. Set **Chart Variables** and **Show Report** to **Yes**.
8. In the Solver Settings dialog box, select **Close**.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

Adams displays the following plots and a design study report:

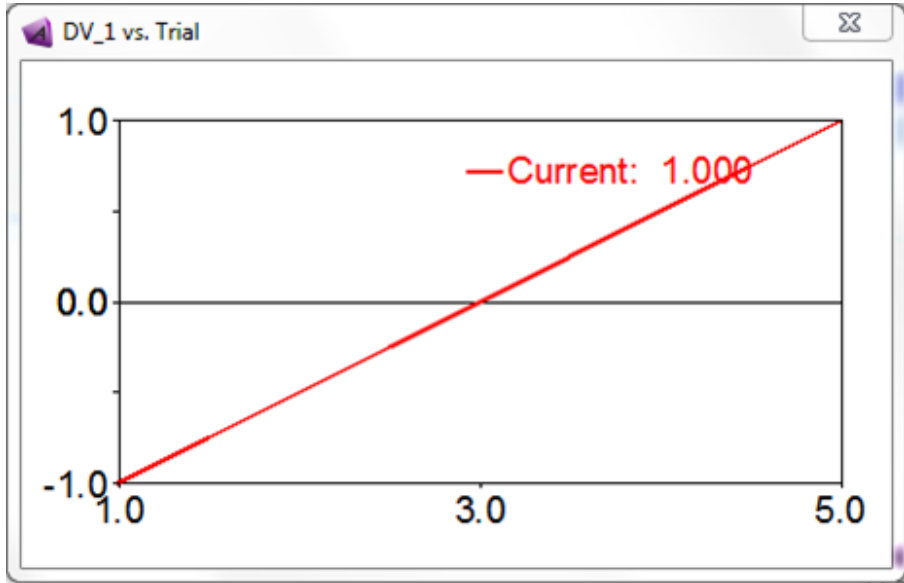
- Spring force plot, with five curves for the five different cases:



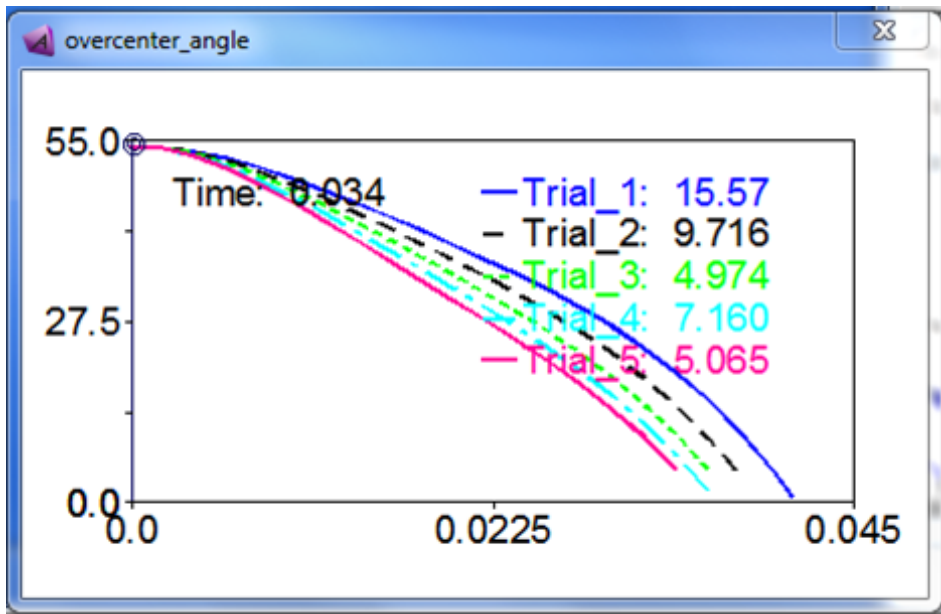
■ SPRING\_1\_force versus DV\_1 plot:



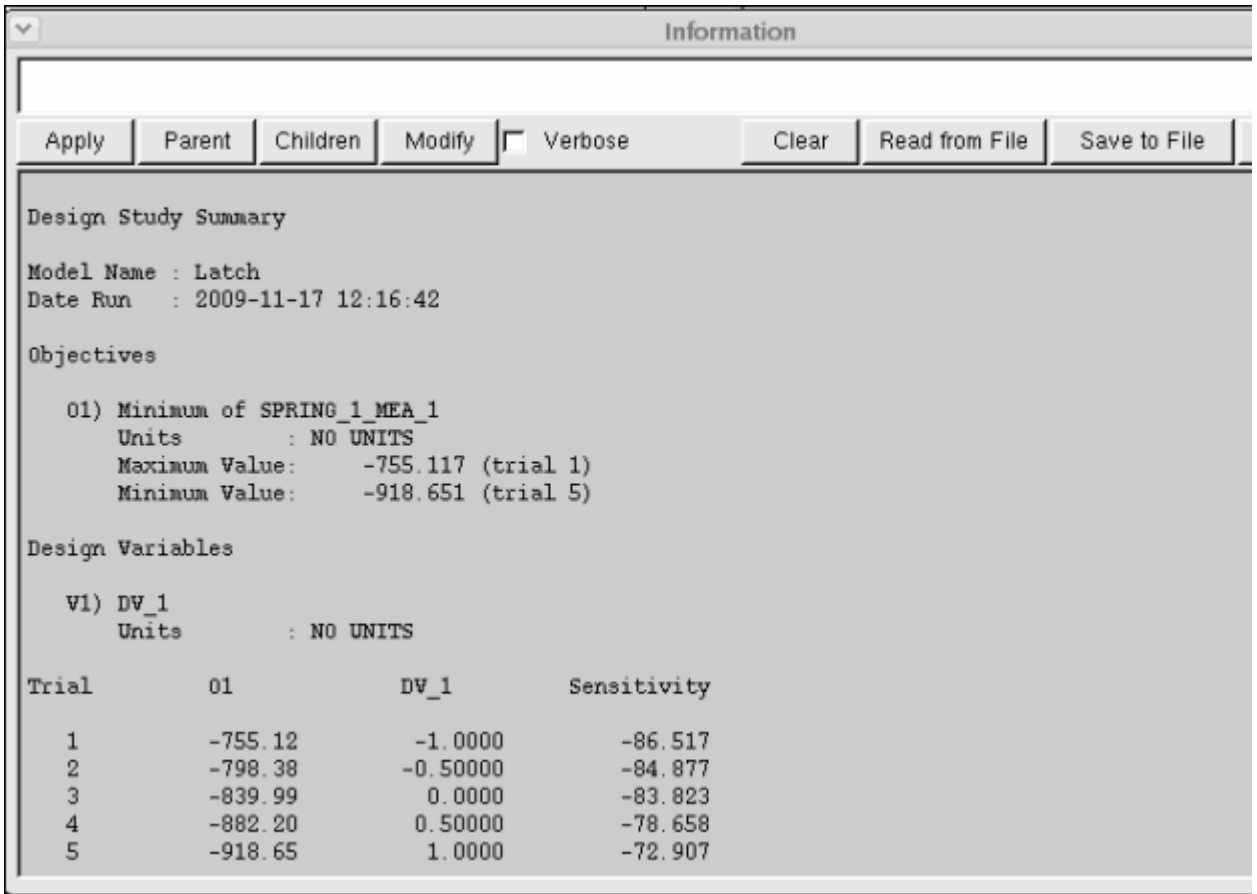
■ DV\_1 versus Trial plot:



- overcenter\_angle plot, with five curves for the five different cases:



- Design study report:



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

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

## Examining the Results of Design Studies

To save your time, we ran some design studies for all the design variables and provided the results in [Table 1](#). 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 1 Design Studies Results

Design variable names:	Design point locations:	Initial value:	Sensitivity at initial value (N/cm):	Apparent optimal value:
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_8_x)	-1	36	-1.1
DV_8	(POINT_8_y)	10	-287	10.5
DV_9	(POINT_8_2_x)	-6	-61	-5.4
DV_10	(POINT_8_2_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.



# 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](#)
- [Running an Optimization](#)



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 1](#) of [Iterating Your Design](#) (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:

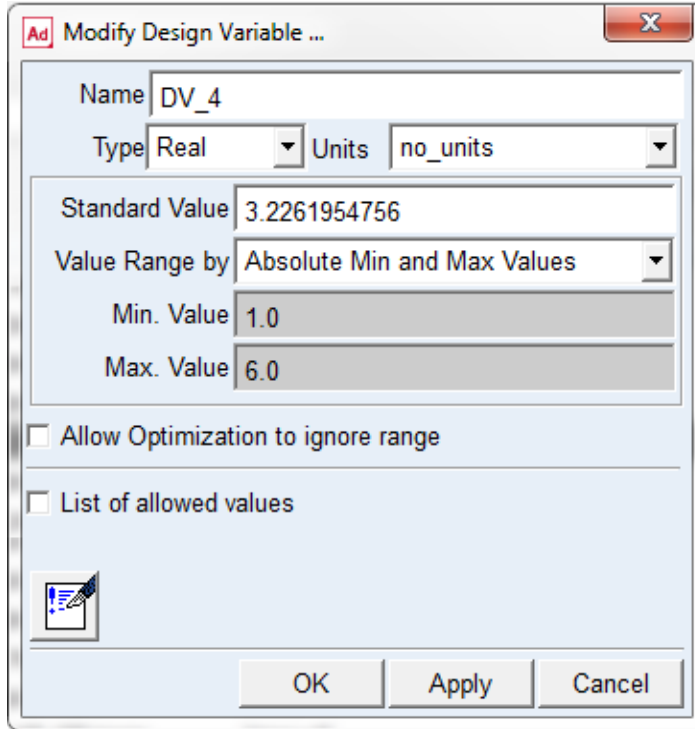
Table 1 Design variable Limits

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_8 y	9	11

### To modify design variables:

1. From the **Model Browser** window, click **Design Variables** and expand.
2. Select the first design variable you need to modify, in this case, DV\_4 from the list. Right click and select **Modify**.

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 1](#).
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 have modified the last design variable, DV\_8, select **OK**.

## Running an Optimization

At this point you will 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. Click the **Design Exploration** tab on the Adams View ribbon.

2. From the **Measures** container, click **Display a Measure** tool  .


The Database Navigator appears.

3. Select **SPRING\_1\_MEA\_1**.
4. Select **OK**.

The **SPRING\_1\_MEA\_1** plot appears.

### To display the overcenter\_angle:

1. Click the **Design Exploration** tab on the Adams View ribbon.

2. From the **Measures** container, click **Display a Measure** tool  .


The Database Navigator appears.

3. Select **overcenter\_angle**.
4. Select **OK**.

5. The **overcenter\_angle** plot appears.

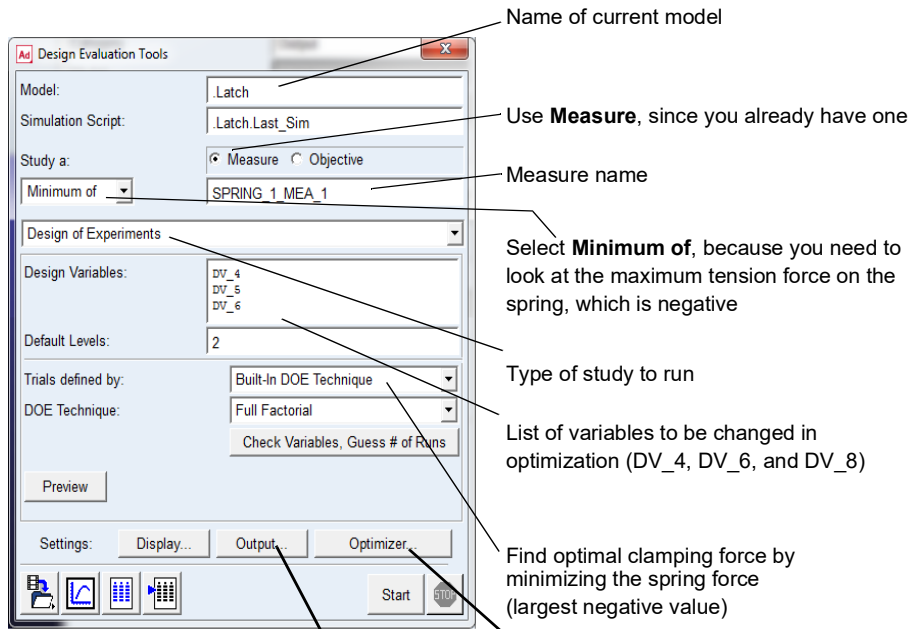
### To run an optimization:

1. Click the **Design Exploration** tab on the Adams View ribbon.

2. From the **Design Evaluation** container, click **Design Evaluation** tool  .

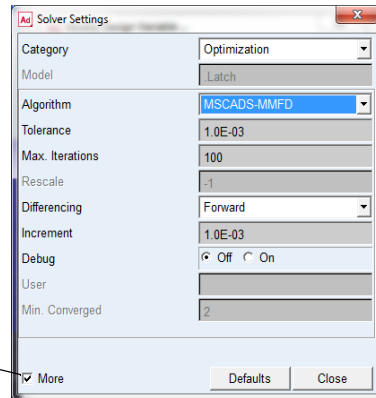
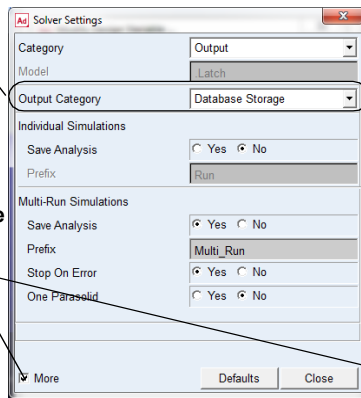
The Design Evaluation Tools dialog box appears.

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



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

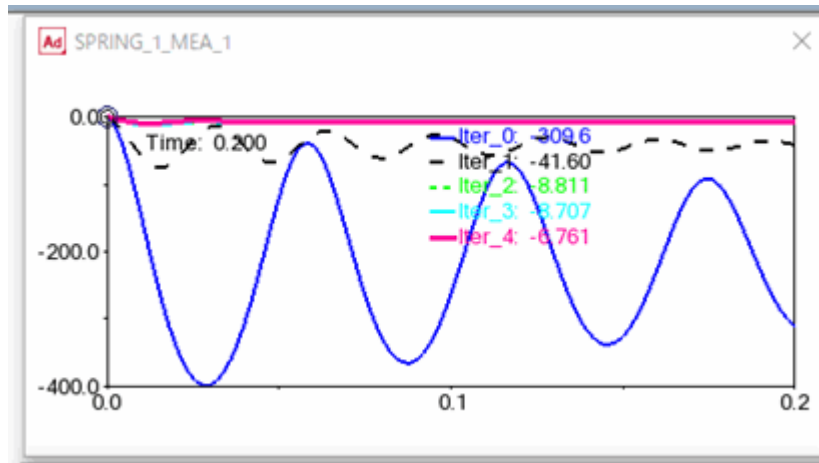
Select **More** to see all options



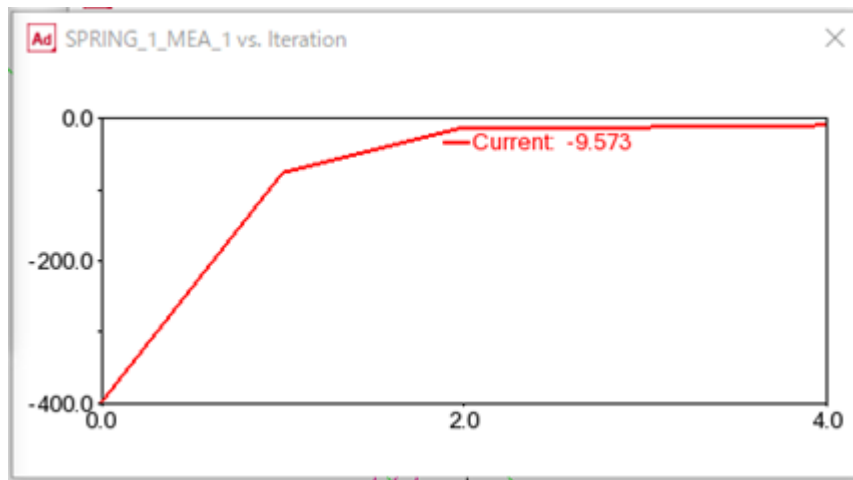
**Note:**

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

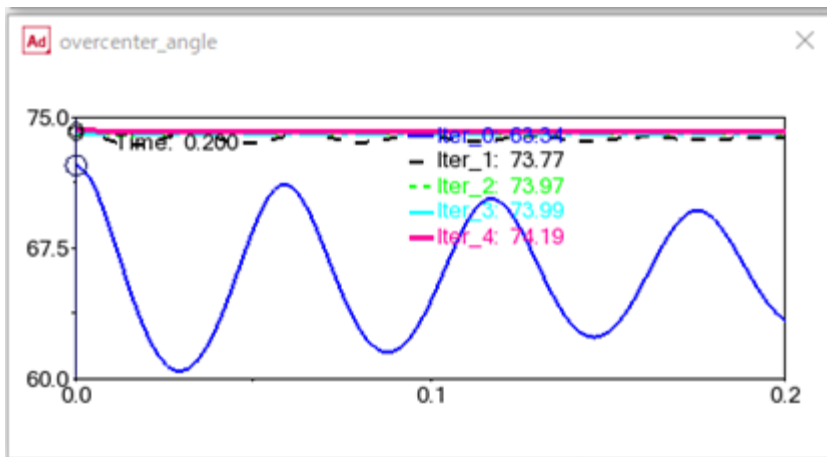
The spring force measure plot shows the optimal clamping force as a function of time. Note that the limits on the strip chart may be different if you worked through the entire tutorial or you loaded a command file and started from a particular chapter.




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.



# 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](#)
- [Creating a Custom Dialog Box](#)
- [Modifying Force Values](#)
- [The Next Step](#)



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. Click the **Design Exploration** tab on the Adams View ribbon.
2. From the **Design Variable** container, click **Design Variable** tool.  
The Create Design Variable dialog box appears.
3. Set **Standard Value** at **80**.
4. Set **Value Range** by to **Absolute Min and Max Values**.
5. Set the minimum value at **60**, and the maximum at **90**.
6. Select **Apply**.
7. Repeat for `DV_12` setting the standard value to **10**, the minimum value to **0**, and the maximum to **20**.
8. 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:

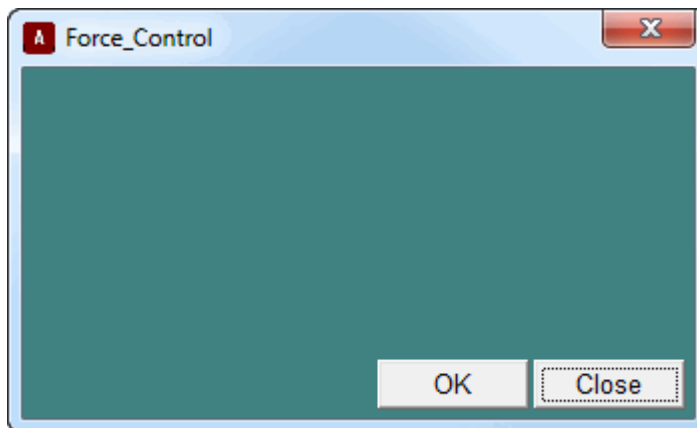
- [Adding Objects to Your Dialog Box](#)
- [Assigning Commands to the Sliders](#)

- [Testing the Dialog Box](#)
- [Saving Your Dialog Box](#)

### 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:

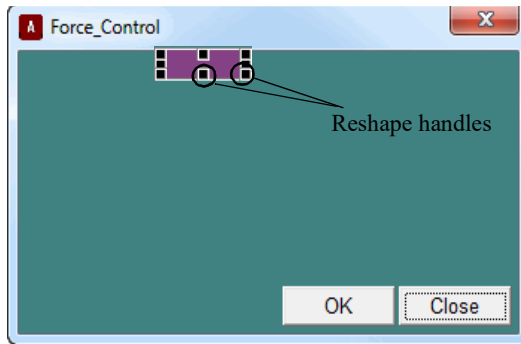


## Adding Objects to Your Dialog Box

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 .
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 .
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.

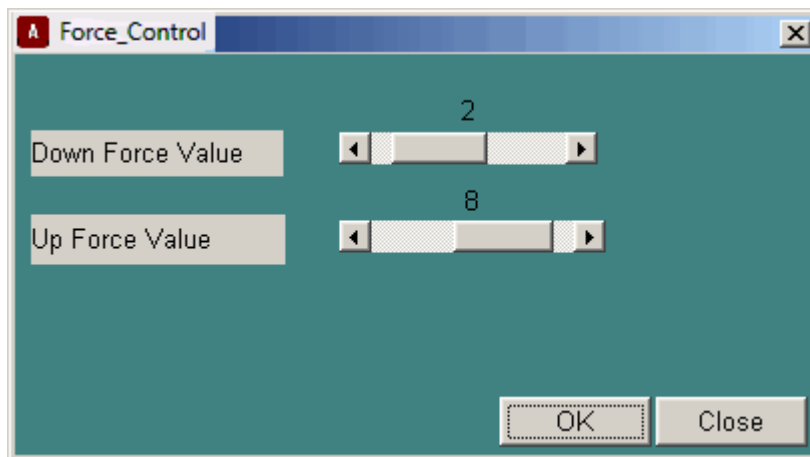


Figure 1 Force Control Dialog Box

## 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** pull-down menu, select **Commands**.
2. At the bottom of the Dialog-Box Builder, select **Execute commands while sliding**.
3. In the command window, enter **variable modify 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 modify 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. Even if you don't save your dialog box to a separate command file, you can still access it through the Database Navigator.

**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 the command file `Force_Control.cmd` in the current working directory.

## 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.

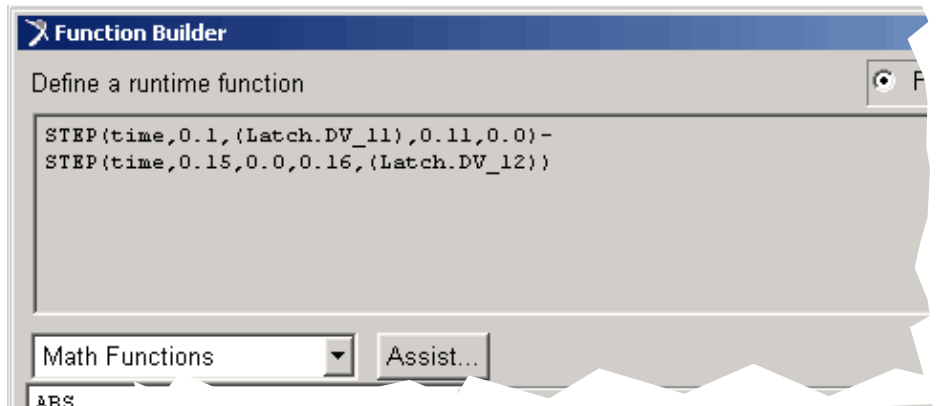
You can try different settings for the clamping and release forces to verify that the handle can be closed and released per the requirements.

**Notes:** If the sensor is causing problems with the investigation, you can deactivate it:  
Click the background to ensure that nothing is selected.

- From the **Edit** menu, select **Deactivate**.
- The Database Navigator appears.
- From the Database Navigator, select the sensor.

**To modify the clamping and release force values:**

1. From the **Model Browser** window, click **Forces** and expand.
2. Select **Force:SFORCE\_1** from the list, right click and select **Modify**.  
The Modify Force dialog box appears.
3. Next to the **Function** text box, select the **More** tool .  
The Function Builder appears.
4. Clear the function text area located at the top of the Function Builder.
5. In the function text area, enter the `STEP` function as shown next:



6. Select **Apply**.

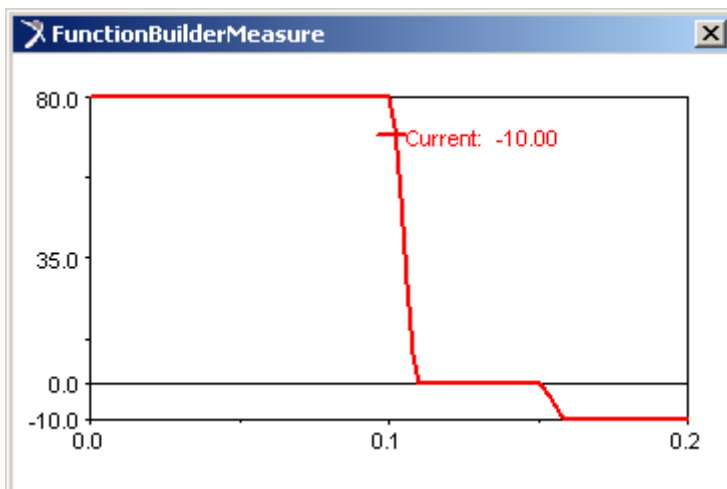
### Plotting the force values:

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 help.

