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Chapter 4
Model History Tree

Learning Objectives

♦ Understand Feature Interactions
♦ Use the Part Browser
♦ Modify and Update Feature Dimensions
♦ Perform History-Based Part Modifications
♦ Change the Names of Created Features
♦ Implement Basic Design Changes
## Autodesk Inventor Certified User Exam Objectives Coverage

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Introduction

In Autodesk Inventor, the design intents are embedded into features in the history tree. The structure of the model history tree resembles that of a CSG binary tree. A CSG binary tree contains only Boolean relations, while the Autodesk Inventor history tree contains all features, including Boolean relations. A history tree is a sequential record of the features used to create the part. This history tree contains the construction steps, plus the rules defining the design intent of each construction operation. In a history tree, each time a new modeling event is created previously defined features can be used to define information such as size, location, and orientation. It is therefore important to think about your modeling strategy before you start creating anything. It is important, but also difficult, to plan ahead for all possible design changes that might occur. This approach in modeling is a major difference of FEATURE-BASED CAD SOFTWARE, such as Autodesk Inventor, from previous generation CAD systems.

Feature-based parametric modeling is a cumulative process. Every time a new feature is added, a new result is created and the feature is also added to the history tree. The database also includes parameters of features that were used to define them. All of this happens automatically as features are created and manipulated. At this point, it is important to understand that all of this information is retained, and modifications are done based on the same input information.

In Autodesk Inventor, the history tree gives information about modeling order and other information about the feature. Part modifications can be accomplished by accessing the features in the history tree. It is therefore important to understand and utilize the feature history tree to modify designs. Autodesk Inventor remembers the history of a part, including all the rules that were used to create it, so that changes can be made to any operation that was performed to create the part. In Autodesk Inventor, to modify a feature we access the feature by selecting the feature in the browser window.
The Saddle Bracket Design

Based on your knowledge of Autodesk Inventor so far, how many features would you use to create the design? Which feature would you choose as the BASE FEATURE, the first solid feature, of the model? What is your choice in arranging the order of the features? Would you organize the features differently if additional fillets were to be added in the design? Take a few minutes to consider these questions and do preliminary planning by sketching on a piece of paper. You are also encouraged to create the model on your own prior to following through the tutorial.

Starting Autodesk Inventor

1. Select the Autodesk Inventor option on the Start menu or select the Autodesk Inventor icon on the desktop to start Autodesk Inventor. The Autodesk Inventor main window will appear on the screen.

2. Once the program is loaded into memory, select the New File icon with a single click of the left-mouse-button in the Launch toolbar.


4. Click Create in the New File dialog box to accept the selected settings to start a new model.
Modeling Strategy
The Autodesk Inventor Browser

- In the Autodesk Inventor screen layout, the browser is located to the left of the graphics window. Autodesk Inventor can be used for part modeling, assembly modeling, part drawings, and assembly presentation. The browser window provides a visual structure of the features, constraints, and attributes that are used to create the part, assembly, or scene. The browser also provides right-click menu access for tasks associated specifically with the part or feature, and it is the primary focus for executing many of the Autodesk Inventor commands.

- The first item displayed in the browser is the name of the part, which is also the file name. By default, the name “Part1” is used when we first started Autodesk Inventor. The browser can also be used to modify parts and assemblies by moving, deleting, or renaming items within the hierarchy. Any changes made in the browser directly affect the part or assembly and the results of the modifications are displayed on the screen instantly. The browser also reports any problems and conflicts during the modification and updating procedure.

Creating the Base Feature

1. Move the graphics cursor to the Start 2D Sketch icon in the Sketch toolbar under the 3D Model tab. A Help-tip box appears next to the cursor and a brief description of the command is displayed at the bottom of the drawing screen.

2. Move the cursor over the edge of the XY Plane in the graphics area. When the XY Plane is highlighted, click once with the left-mouse-button to select the Plane as the sketch plane for the new sketch.
3. Select the **Line** icon by clicking once with the left-mouse-button; this will activate the **Line** command.

4. On your own, create and adjust the geometry by adding and modifying dimensions as shown below.

![Geometry Diagram]

5. Inside the graphics window, click once with the right-mouse-button to display the option menu. Select **Finish 2D Sketch** in the pop-up menu to end the **Sketch** option.

6. On your own, use the dynamic viewing functions to view the sketch. Click the home view icon to change the display to the **isometric** view.

7. In the **Sketch toolbar** under the **3D Model tab**, select the **Extrude** command by left-clicking on the icon.
8. In the *Distance* option box, enter **2.5** as the total extrusion distance.

9. In the *Extrude* pop-up window, left-click once on the **Symmetric** icon. The **Symmetric** option allows us to extrude in both directions of the sketched profile.

10. Click on the **OK** button to accept the settings and create the base feature.

➢ On your own, use the *Dynamic Viewing* functions to view the 3D model. Also notice the extrusion feature is added to the *Model Tree* in the *browser* area.
Adding the Second Solid Feature

1. In the Sketch toolbar under the 3D Model tab select the Start 2D Sketch command by left-clicking once on the icon.

2. In the Status Bar area, the message “Select plane to create sketch or an existing sketch to edit.” is displayed. Move the graphics cursor on the 3D part and notice that Autodesk Inventor will automatically highlight feasible planes and surfaces as the cursor is on top of the different surfaces. Move the cursor inside the upper horizontal face of the 3D object as shown below.

3. Click once with the right-mouse-button to bring up the option menu and choose Select Other to switch to the next feasible choice.

4. On your own, click on the down arrow to examine all possible surface selections.

5. Select the bottom horizontal face of the solid model when it is highlighted as shown in the above figure.
Creating a 2D Sketch

1. Select the **Center Point Circle** command by clicking once with the left-mouse-button on the icon in the **Sketch** tab.

   ➢ We will align the center of the circle to the midpoint of the base feature.

2. On your own, use the snap to midpoint option to pick the midpoint of the edge when the midpoint is displayed with GREEN color as shown in the figure. (Hit [F6] to set the display orientation if necessary.)

3. Select the front corner of the base feature to create a circle as shown below.

4. Inside the graphics window, click once with the right-mouse-button to display the option menu. Select **OK** in the pop-up menu to end the **Circle** command.
5. Inside the graphics window, click once with the right-mouse-button to display the option menu. Select **Finish 2D Sketch** in the pop-up menu to end the Sketch option.

6. In the **Features** toolbar (the toolbar that is located to the left side of the graphics window), select the **Extrude** command by clicking the left-mouse-button on the icon.

7. Click inside the circle we just created and left-click once to select the region as the profile to be extruded.

8. In the **Extrude** pop-up control, set the operation option to **Join**.

9. Also set the Extents option to **To Selected Face** as shown below.

10. Select the top face of the base feature as the termination surface for the extrusion.

11. Click on the OK button to proceed with the Join operation.
Renaming the Part Features

Currently, our model contains two extruded features. The feature is highlighted in the display area when we select the feature in the browser window. Each time a new feature is created, the feature is also displayed in the Model Tree window. By default, Autodesk Inventor will use generic names for part features. However, when we begin to deal with parts with a large number of features, it will be much easier to identify the features using more meaningful names. Two methods can be used to rename the features: 1. **Clicking** twice on the name of the feature and 2. Using the **Properties** option. In this example, the use of the first method is illustrated.

1. Select the first extruded feature in the **model browser** area by left-clicking once on the name of the feature, **Extrusion1**. Notice the selected feature is highlighted in the graphics window.

2. Left-click again on the feature name to enter the **Edit** mode as shown.

3. Enter **Base** as the new name for the first extruded feature.

4. On your own, rename the second extruded feature to **Circular_End**.
Adjusting the Width of the Base Feature

One of the main advantages of parametric modeling is the ease of performing part modifications at any time in the design process. Part modifications can be done through accessing the features in the history tree. Autodesk Inventor remembers the history of a part, including all the rules that were used to create it, so that changes can be made to any operation that was performed to create the part. For our Saddle Bracket design, we will reduce the size of the base feature from 3.25 inches to 3.0 inches, and the extrusion distance to 2.0 inches.

1. Select the first extruded feature, Base, in the browser area. Notice the selected feature is highlighted in the graphics window.

2. Inside the browser area, right-mouse-click on the first extruded feature to bring up the option menu and select the Show Dimensions option in the pop-up menu.

3. All dimensions used to create the Base feature are displayed on the screen. Select the overall width of the Base feature, the 3.25 dimension value, by double-clicking on the dimension text as shown.

4. Enter 3.0 in the Edit Dimension window.

5. On your own, repeat the above steps and modify the extruded distance from 2.5 to 2.0.

6. Click Local Update in the Quick Access Toolbar.

   Note that Autodesk Inventor updates the model by re-linking all elements used to create the model. Any problems or conflicts that occur will also be displayed during the updating process.
Adding a Placed Feature

1. In the Sketch tab, select the **Hole** command by left-clicking on the icon.

2. In the **Hole** dialog box, choose **Concentric** in the placement option as shown.

3. Pick the **bottom plane** of the solid model as the placement plane as shown.

4. Pick the **bottom arc** to use as the *concentric* reference.

5. Set the hole diameter to **0.75 in** as shown.
6. Set the termination option to **Through All** as shown.

7. Click **OK** to accept the settings and create the **Hole** feature.
Creating a Rectangular Cut Feature

1. In the Sketch toolbar under the 3D Model tab select the Start 2D Sketch command by left-clicking once on the icon.

2. Pick the vertical face of the solid as shown. (Note the alignment of the origin of the sketch plane.)

- On your own, create a rectangular cut (1.0 x 0.75) feature (To Next option) as shown and rename the feature to Rect_Cut.
History-Based Part Modifications

Autodesk Inventor uses the *history-based part modification* approach, which enables us to make modifications to the appropriate features and re-link the rest of the history tree without having to reconstruct the model from scratch. We can think of it as going back in time and modifying some aspects of the modeling steps used to create the part. We can modify any feature that we have created. As an example, we will adjust the depth of the rectangular cutout.

1. In the *browser* window, select the last cut feature, **Rect_Cut**, by left-clicking once on the name of the feature.

2. In the *browser* window, right-click once on the **Rect_Cut** feature.

3. Select **Edit Feature** in the pop-up menu. Notice the *Extrude* dialog box appears on the screen.

4. In the *Extrude* dialog box, set the termination *Extents* to the **Through All** option.

5. Click on the **OK** button to accept the settings.

- As can been seen, the history-based modification approach is very straightforward and it only took a few seconds to adjust the cut feature to the **Through All** option.
A Design Change

Engineering designs usually go through many revisions and changes. Autodesk Inventor provides an assortment of tools to handle design changes quickly and effectively. We will demonstrate some of the tools available by changing the Base feature of the design.

1. In the browser window, select the Base feature by left-clicking once on the name of the feature.

2. In the browser, right-click once on the Base feature to bring up the option menu; then pick Edit Sketch in the pop-up menu.

3. Click Home to reset the display to Isometric.

Autodesk Inventor will now display the original 2D sketch of the selected feature in the graphics window. We have literally gone back to the point where we first created the 2D sketch. Notice the feature being modified is also highlighted in the desktop browser.

4. Click on the Look At icon in the Standard toolbar area.
   - The Look At command automatically aligns the sketch plane of a selected entity to the screen.

5. Select any line segment of the 2D sketch to reset the display to align to the 2D sketch.
6. Select the **Fillet** command in the 2D Sketch toolbar.

7. In the graphics window, enter **0.25** as the new radius of the fillet.

8. Select the two edges as shown to create the fillet.

- Note that the fillet is created automatically with the dimension attached. The attached dimension can also be modified through the history tree.

9. Click on the [X] icon in the 2D Fillet window to end the Fillet command.

10. Select **Finish Sketch** in the Ribbon toolbar to end the Sketch option.
In a typical design process, the initial design will undergo many analyses, testing, and reviews. The *history-based part modification* approach is an extremely powerful tool that enables us to quickly update the design. At the same time, it is quite clear that PLANNING AHEAD is also important in doing feature-based modeling.

11. In the *Standard* toolbar, click **Save** and save the model as *Saddle_Bracket.ipt*. (On your own, create the Chapter4 folder.)
Assigning and Calculating the Associated Physical Properties

Autodesk Inventor models have properties called iProperties. The iProperties can be used to create reports and update assembly bills of materials, drawing parts lists, and other information. With iProperties, we can also set and calculate physical properties for a part or assembly using the material library. This allows us to examine the physical properties of the model, such as weight or center of gravity.

1. In the browser, right-click once on the part name to bring up the option menu; then pick iProperties in the pop-up menu.

2. On your own, look at the different information listed in the iProperties dialog box.

3. Click on the Physical tab; this is the page that contains the physical properties of the selected model.

- Note that the Material option is not assigned, and none of the physical properties are shown.

4. Click the down-arrow in the Material option to display the material list, and select Aluminum-6061 as shown.
5. Click on the **Global** button to display the *Mass Moments Inertial* of the design.

6. On your own, select **Cast Iron** as the *Material* type and compare the differences in using the different materials.

- Also note the *Material* can be assigned through the quick access menu as shown.
Review Questions: (Time: 30 minutes.)

1. What are stored in the Autodesk Inventor History Tree?

2. When extruding, what is the difference between Distance and Through All?

3. Describe the history-based part modification approach.

4. What determines how a model reacts when other features in the model change?

5. Describe the steps to rename existing features.

6. Describe two methods available in Autodesk Inventor to modify the dimension values of parametric sketches.

7. Create History Tree sketches showing the steps you plan to use to create the two models shown on the next page:

Ex.1)

Ex.2)
Exercises: Create and save the exercises in the Chapter4 folder. (Time: 180 minutes.)

1. C-Clip (Dimensions are in inches. Plate thickness: 0.25 inches.)

2. Tube Mount (Dimensions are in inches.)
3. **Hanger Jaw** (Dimensions are in inches. Volume =\?)

4. **Transfer Fork** (Dimensions are in inches. Material: **Cast Iron**. Volume =\?)
5. **Guide Slider** (Material: **Cast Iron**, Weight and Volume =?)

![Guide Slider Diagram]


![Shaft Guide Diagram]
Parametric Modeling with Autodesk® Inventor® 2018

Randy H. Shih

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