



Chapter 1

Introduction to CATIA V5

Learning Objectives

After completing this chapter, you will be able to:

- *Understand the benefits of using CATIA V5.*
- *Understand various workbenches of CATIA V5.*
- *Understand important terms and definitions used in CATIA V5.*
- *Understand the system requirements to install CATIA V5.*
- *Understand the functions of the mouse buttons.*
- *Understand the use of Hot Keys.*
- *Modify the color scheme in CATIA V5.*

INTRODUCTION TO CATIA V5R20

Welcome to **CATIA (Computer Aided Three Dimensional Interactive Application)**. As a new user of this software package, you will join hands with thousands of users of this high-end CAD/CAM/CAE tool worldwide. If you are already familiar with the previous releases, you can upgrade your designing skills with the tremendous improvement in this latest release.

CATIA V5, developed by Dassault Systemes, France, is a completely re-engineered, next-generation family of CAD/CAM/CAE software solutions for Product Lifecycle Management. Through its exceptionally easy-to-use and state-of-the-art user interface, CATIA V5 delivers innovative technologies for maximum productivity and creativity, from the inception concept to the final product. CATIA V5 reduces the learning curve, as it allows the flexibility of using feature-based and parametric designs.

CATIA V5 provides three basic platforms: P1, P2, and P3. P1 is for small and medium-sized process-oriented companies that wish to grow toward the large scale digitized product definition. P2 is for the advanced design engineering companies that require product, process, and resource modeling. P3 is for the high-end design applications and is basically for Automotive and Aerospace Industry, where high quality surfacing or Class-A surfacing is used.

The subject of interpretability offered by CATIA V5 includes receiving legacy data from the other CAD systems and even between its own product data management modules. The real benefit is that the links remain associative. As a result, any change made to this external data gets notified and the model can be updated quickly.

CATIA V5 WORKBENCHES

CATIA V5 serves the basic design tasks by providing different workbenches. A workbench is defined as a specified environment consisting of a set of tools that allows the user to perform specific design tasks. The basic workbenches in CATIA V5 are **Part Design**, **Wireframe and Surface Design**, **Assembly Design**, **Drafting**, **Generative Sheetmetal Design**, and **DMU Kinematics**. These workbenches are discussed next.

Part Design Workbench

The **Part Design** workbench is a parametric and feature-based environment in which you can create solid models. The basic requirement for creating a solid model in this workbench is a sketch. The sketch for the features is drawn in the **Sketcher** workbench that can be invoked within the **Part Design** workbench. You can draw the sketch using the tools in this workbench. While drawing a sketch, some constraints are automatically applied to it. You can also apply additional constraints and dimensions manually. After drawing the sketch, exit the **Sketcher** workbench and convert it into a feature. The tools in the **Part Design** workbench can be used to convert the sketch into a sketch-based feature. This workbench also provides other tools to apply the placed features, such as fillets, chamfers, and so on. These features are called the dress-up features. You can also assign materials to the model in this workbench.

Wireframe and Surface Design Workbench

The **Wireframe and Surface Design** workbench is also a parametric and feature-based

environment, and is used to create wireframe or surface models. The tools in this workbench are similar to those in the **Part Design** workbench with the only difference that the tools in this environment are used to create basic and advanced surfaces.

Assembly Design Workbench

The **Assembly Design** workbench is used to assemble the components using the assembly constraints available in this workbench. There are two types of assembly design approaches:

1. Bottom-up
2. Top-down

In the bottom-up approach of the assembly design, the components are assembled together to maintain their design intent. In the top-down approach, components are created inside the assembly in the **Assembly Design** workbench. You can also assemble an existing assembly to the current assembly. The **Space Analysis** toolbar provides the **Clash** analysis tool that helps in detecting clash, clearance, and contact between components and subassemblies.

Drafting Workbench

The **Drafting** workbench is used for the documentation of the parts or assemblies created earlier in the form of drawing views and their detailing. There are two types of drafting techniques:

1. Generative drafting
2. Interactive drafting

The generative drafting technique is used to automatically generate the drawing views of the parts and assemblies. The parametric dimensions added to the component in the **Part Design** workbench during its creation can be generated and displayed automatically in the drawing views. The generative drafting is bidirectionally associative in nature. You can also generate the Bill of Material (BOM) and balloons in the drawing views.

In interactive drafting, you need to create the drawing views by sketching them using the normal sketching tools and then adding the dimensions.

Generative Sheetmetal Design Workbench

The **Generative Sheetmetal Design** workbench is used for the designing the sheet metal components. Generally, the solid models of the sheet metal components are created to generate the flat pattern of the sheet, study the design of the dies and punches, study the process plan for designing, and the tools needed for manufacturing the sheet metal components.

DMU Kinematics Workbench

The **DMU Kinematics** workbench is used to design mechanisms by adding joints between the components. It also allows you to simulate and analyze the working of the mechanisms dynamically.

SYSTEM REQUIREMENTS

The following are the system requirements to ensure the smooth running of CATIA V5R20 on your system:

- System unit: An Intel Pentium 4 or Xeon-based workstation running Microsoft 2000 Professional Edition or Windows XP or Windows Vista or Windows 7.
- Memory: 512 MB of RAM (minimum).
- Disk drive: 4 GB Disk Drive space (Minimum recommended size)
- Internal/External drives: A CD-ROM drive for installing programs.
- Display: A graphic color display compatible with the selected platform-specific graphic adapter. The minimum recommended monitor size is 17 inches.
- Graphics adapter: A graphics adapter with a 3D OpenGL accelerator with a minimum resolution of 1024x768 pixels for Microsoft Windows workstations and 1280x1024 for UNIX workstations.

GETTING STARTED WITH CATIA V5R20

Install CATIA V5R20 on your system and then start it by double-clicking on the shortcut icon of **CATIA V5R20** on the desktop of your computer. You can also choose **Start > All Programs > CATIA > CATIA V5R20** from the taskbar to start the program, as shown in Figure 1-1.

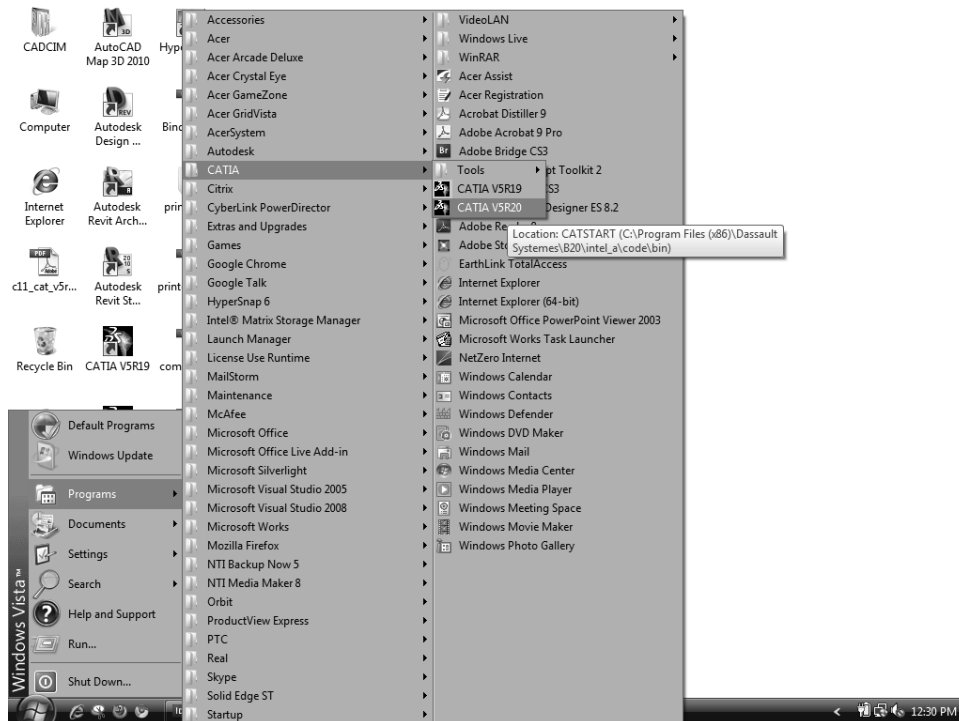


Figure 1-1 Starting CATIA V5R20 from the Start menu

After the system has loaded all required files to start CATIA V5R20, a new product file, with the default name **Product1** will start automatically, as shown in Figure 1-2.

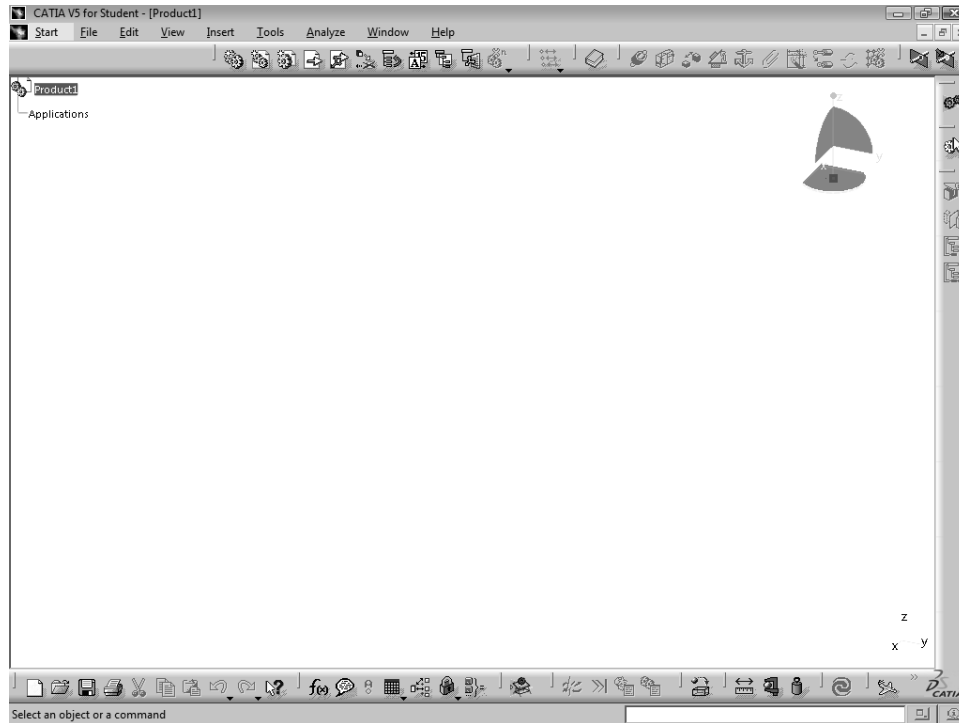


Figure 1-2 The initial screen that appears after starting CATIA V5R20

Close this file by choosing **File > Close** from the menu bar. Figure 1-3 shows the screen that appears after closing the initial Product file.

IMPORTANT TERMS AND DEFINITIONS

Some important terms and definitions used in CATIA V5R20 are discussed next.

Feature-based Modeling

A feature is defined as the smallest building block that can be modified individually. A model created in CATIA V5 is a combination of a number of directly or indirectly related individual features. You can modify these features any time during the design process. If proper design intent is maintained while creating the model, then these features automatically adjust according to the change occurred in their surroundings. This provides greater flexibility to the design.

Parametric Modeling

The parametric nature of a software package is defined as its ability to use the standard properties or parameters in defining the shape and size of a geometry. The main function of this property is to transform the selected geometry to a new size or shape without considering its original dimensions. You can change or modify the shape and size of any feature at any

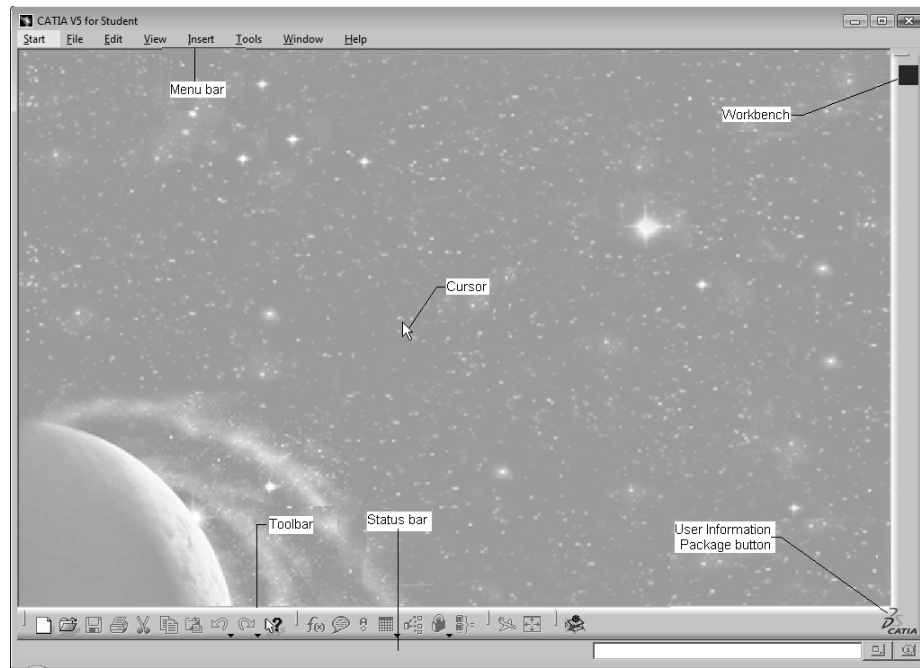


Figure 1-3 The screen that appears after closing the initial Product file

stage of the design process. This property makes the designing process very easy. For example, consider the design of the body of a pipe housing shown in Figure 1-4.

To change the design by modifying the diameter of the holes and their number on the front, top, and bottom faces, you have to simply select the feature and change the diameter and the number of instances in the pattern. The modified design is shown in Figure 1-5.

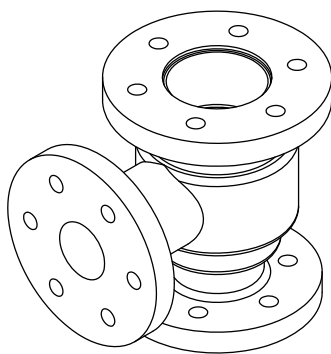


Figure 1-4 Body of a pipe housing

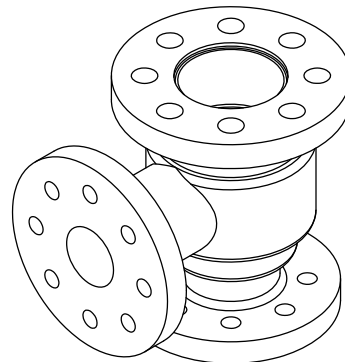


Figure 1-5 Modified body of a pipe housing

Bidirectional Associativity

As mentioned earlier, CATIA V5 has different workbenches such as **Part Design**, **Assembly Design**, **Drawing**, and **Generative Sheetmetal Design**. The bidirectional associativity that exists between all these workbenches ensures that any modification made in the model in any of the workbenches of CATIA V5, reflects automatically and immediately in other workbenches also. For example, if you modify the dimension of a part in the **Part Design** workbench, the change will reflect in the **Assembly Design** and **Drawing** workbenches also. Similarly, if you modify the dimensions of a part in the drawing views generated in the **Drawing** workbench, the changes will reflect in the **Part Design** and **Assembly Design** workbenches also. Consider the drawing views of the pipe housing shown in Figure 1-6. When you modify the model in the **Part Design** workbench, the changes are reflected in the **Drawing** workbench automatically. Figure 1-7 shows the drawing views of the pipe housing after increasing the diameter and number of holes at the top flange.

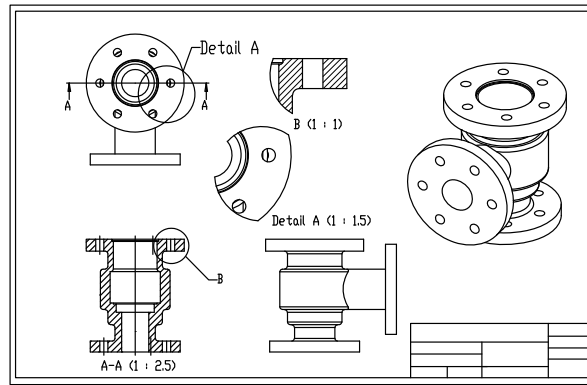


Figure 1-6 The drawing views of the pipe housing before making modifications

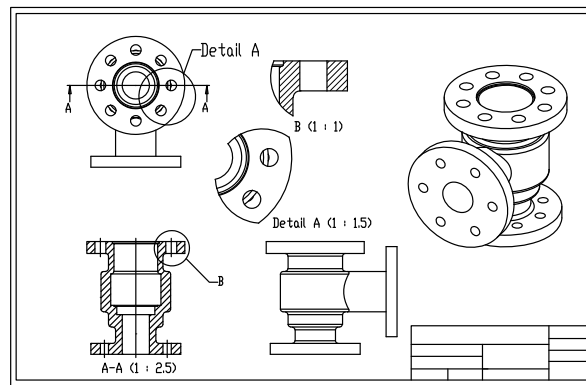


Figure 1-7 The drawing views of the pipe housing after modifications

CATPart

CATPart is a file extension associated with all those files that are created in the **Sketcher**, **Part Design**, **Generative Sheetmetal Design**, and **Wireframe and Surface Design** workbenches of CATIA V5.

CATProduct

CATProduct is a file extension associated with all those files that are created in the **Assembly Design** workbench of CATIA V5.

CATDrawing

CATDrawing is a file extension associated with all those files that are created in the **Drafting** workbench of CATIA V5.

Specification Tree

The Specification tree keeps track of all operations carried out on the part. Figure 1-8 shows the specification tree that appears when you start a new file under the **Part Design** workbench.

Compass

The compass is used to manipulate the orientation of parts, assemblies, or sketches. You can also orient the view of the parts and assemblies. The compass is shown in Figure 1-9. By default, it appears on the top right corner of the geometry area.



Figure 1-8 The Specification tree that appears on starting a new **CATPart** file



Figure 1-9 The Compass

Constraints

Constraints are the logical operations that are performed on the selected element to define its size and location with respect to the other elements or reference geometries. There are two types of constraints in CATIA V5. The constraints in the **Sketcher** workbench are called geometric constraints and are used to precisely define the size and position of the sketched elements with respect to the surroundings. The assembly constraints available in the **Assembly Design** workbench are used to define the precise position of the components in the assembly. These constraints are discussed next.

Geometric Constraints

Geometric constraints are the logical operations performed on the sketched elements to define their size and position with respect to the other elements. There are two methods to apply Geometric constraints, automatic and manual. While drawing the sketch, some constraints are automatically applied to it. For applying constraints manually, you need to invoke the **Constraints Defined in Dialog Box** tool and select the appropriate check box.

The constraints in the **Sketcher** workbench are discussed next.

Distance

This constraint is used to apply a distance dimension between any two selected entities.

Length

This constraint is used to apply a linear dimension to the selected line.

Angle

This constraint is used to apply an angular dimension between any two selected lines.

Radius / Diameter

This constraint is used to apply a radius or diameter dimension to the selected circular entity.

Semimajor axis

This constraint is used to apply a dimension to the major axis of the selected ellipse.

Semiminor axis

This constraint is used to apply a dimension to the minor axis of the selected ellipse.

Symmetry

This constraint is used to force the selected sketched entities symmetrical about an axis. A line segment can be used as an axis.

Midpoint

This constraint forces a selected point to be placed on the midpoint of the selected line.

Equidistant point

This constraint forces a selected point to be placed at an equal distance from any two preselected points.

Fix

This constraint is used to fix a selected entity in terms of its position with respect to the coordinate system of the current sketch.

Coincidence

This constraint is used to make two points, two lines, a point and a line, or a point and a curve coincident.

Concentricity

This constraint is used to make two circles, arc, an arc and a circle, a point and a circle, or a point and an arc concentric.

Tangency

This constraint is used to force the selected line segment or curve to become tangent to another curve.

Parallelism

The **Parallelism** constraint is used to force any two selected line segments to become parallel to each other. The selected line segments can be axes also.

Perpendicular

The **Perpendicular** constraint is used to force any two selected line segments to become perpendicular to each other. The selected line segments can be axes also.

Horizontal

The **Horizontal** constraint forces the selected line segment to become horizontal.

Vertical

The **Vertical** constraint forces the selected line segment to become vertical.

Assembly Constraints

The constraints in the **Assembly Design** workbench are the logical operations performed to restrict the degrees of freedom of a component and to define its precise location and position with respect to the other components in the assembly. The constraints in this workbench are discussed next.

Coincidence

This constraint is used to force two selected entities to coincide with each other. The selected entities can be central axes of circular components, two adjacent or opposite faces, or two adjacent planes.

Contact

This constraint is used to force two selected faces to maintain contact with each other.

Offset

This constraint is used to place two different selected faces, planes, or central axes at a distance with respect to each other.

Angle

This constraint is used to place two selected entities at an angle with respect to each other. These entities can be the central axes of circular components, two faces, two planes, or a combination of an axis and a face, a plane and a face, or an axis and a plane.

Fix

This constraint fixes the position of the selected part in the 3D space.

Fix Together

This constraint fixes the position of two different selected parts with respect to each other.

Quick

The **Quick Constraint** tool is used to apply the most appropriate constraint to the elements in the current selection set. You can set the priority depending on which CATIA V5 will perform the constraint selection.

PartBody

The PartBody is the default body in the **Part Design** workbench. All solid related features, such as pad, pocket, shaft, and so on are placed inside it. Other bodies that will be inserted under the **Part Design** workbench will be named as **Body.2**, **Body.3**, and so on.

Geometrical Set

The geometrical set is defined as a body that includes the newly created planes, surfaces, wireframe elements, and reference elements.

Wireframe

The wireframe construction elements aid in creating surfaces. They generally consist of points, lines, and arcs, and are used as substitutes of entities drawn in the **Sketcher** workbench.

Surface

Surfaces are geometric features which have no thickness. They are used to create complex shapes that are difficult to make using solid features. After creating a surface, you can assign a thickness to it to convert it into a solid body.

Feature

A feature is defined as a basic building block of a solid model. The combination of various features results in a complete model. In the **Part Design** workbench of CATIA V5, the features are of the following four types:

1. Sketch-Based Features
2. Dress-Up Features
3. Transformation Features
4. Surface-Based Features

Reframe on

Sometimes, a feature, a body, or a sketch may not be visible in the available space of the geometry area. The **Reframe on** option is available in the contextual menu is used to view the particular selection in the available display space.

Center Graph

The **Center graph** option available in the contextual menu is used to bring the selected feature, body, or sketch in the Specification tree to the middle left portion of the geometry area.

UNDERSTANDING THE FUNCTIONS OF THE MOUSE BUTTONS

To work in CATIA V5 design workbenches, it is necessary that you understand the functions of the mouse buttons. The efficient use of these three buttons, along with the CTRL key on the keyboard, can reduce the time required to complete the design task. The different combinations of the CTRL key and mouse buttons are listed below:

1. The left mouse button is used to make a selection by dragging a window or by simply selecting a face, surface, sketch, or object from the geometry area or from the Specification tree. For multiple selections, press and hold the CTRL key and select the entities using the left mouse button.
2. The right mouse button is used to invoke the contextual (shortcut) menu after selecting an element or invoking a tool.
3. Press and hold the middle mouse button and drag the mouse to pan the view of the model on the screen.
4. Press and hold the middle mouse button and then click the right mouse button once to invoke the **Zoom** mode. Now, drag the mouse up to zoom in the view of the model. Similarly, drag it down to zoom out the view of the model. You can also invoke the zoom tool by first pressing and holding the CTRL key and then pressing and holding the middle mouse button. Now, release the CTRL key and drag the cursor to zoom in and out of the view of the model. Figure 1-10 shows how to use a three-button mouse to perform the zoom in and zoom out functions.

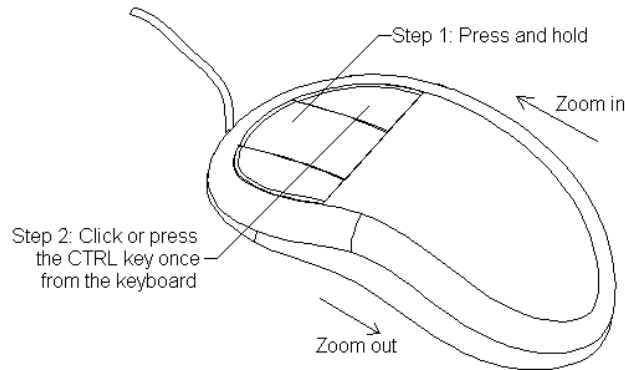


Figure 1-10 Using the three-button mouse to perform the zoom in and zoom out operations

5. Press and hold the middle mouse button. Then, press and hold the right mouse button to invoke the **Rotate** mode. Next, drag the mouse to dynamically rotate the view of the model in the geometry area and view it from different directions. You can also invoke the rotate view tool by pressing and holding the middle mouse button and then pressing and holding the CTRL key. Next, drag the cursor to rotate the view of the model. Figure 1-11 shows how to use the three-button mouse to perform the rotate operation.



Note

It is assumed that a three-button mouse is configured on your system.

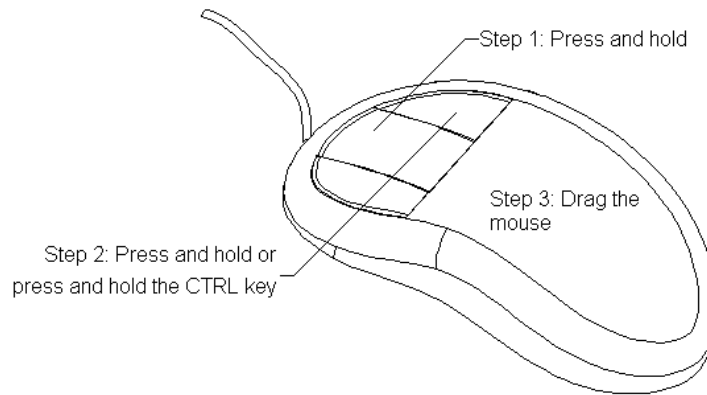


Figure 1-11 Using the three-button mouse to perform the rotate operation

TOOLBARS

CATIA V5 offers a user-friendly design environment by providing specific toolbars to each workbench. Therefore, it is important that you get acquainted with various standard toolbars and buttons that appear in the workbenches of CATIA V5. These toolbars are discussed next.

Standard Toolbar

This toolbar is common to all workbenches of CATIA V5. Figure 1-12 shows the **Standard** toolbar.

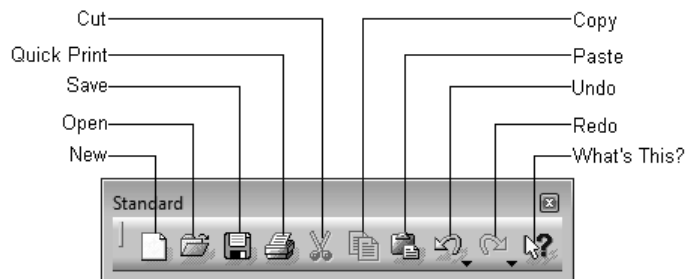


Figure 1-12 The **Standard** toolbar

The buttons in this toolbar are used to start a new file, open an existing file, save a file, and print the current document. These buttons are also used to cut and place the selection on a temporary clipboard, copy a selection, paste the content from the clipboard to a selected location, undo, redo, and invoke the help topics. The **What's This?** button provides help on the toolbar icons.

Status Bar

The status bar, which appears at the bottom of the CATIA V5 window, comprises of three areas, as shown in Figure 1-13. These areas are discussed next.



Figure 1-13 The status bar

Current Information or Dialog Box

The **Current Information or Dialog Box** area displays the current information about the selected feature or current tool.

Power Input Field Bar

The **Power Input Field** bar lets you invoke the commands and enter the data or value that can be directly associated with the feature.



Note

*In case an incorrect command is entered in the **Power Input Field** bar, a **Power input message** dialog box appears indicating the unknown command or syntax error. Choose the **OK** button from this dialog box.*

*To launch any command using the **Power Input Field** bar, the general syntax of command will be C: <name of the command>. For example, to start a new file, enter C: New.*



Tip. *It is recommended that you invoke the tools from the toolbars or menu bar, instead of using the **Power Input Field** bar, because entering commands in the **Power Input Field** box is a tedious process.*

Dialog Box Display Button

Choosing the **Dialog Box Display** button will turn on or off the display of the current dialog box.

Part Design Workbench Toolbars

You can invoke the **Part Design** workbench by choosing the **New** button from the **Standard** toolbar and selecting **Part** from the **New** dialog box. Alternatively, you can choose **Start > Mechanical Design > Part Design** from the menu bar. The toolbars in the **Part Design** workbench are discussed next.

View Toolbar

The buttons in the **View** toolbar, shown in Figure 1-14, are used for manipulating the view of the model using the tools such as pan, zoom, normal viewing about a planar surface, face or plane, defining a render style, and so on. The **View** toolbar is available in all the workbenches.



Note

*Buttons such as **Fly Mode**, **Normal View**, **Isometric View**, **View Mode**, and **Rotate** are not available in the **Drafting** workbench. For all other workbenches discussed in this book, the*

function of the **View** toolbar is the same.

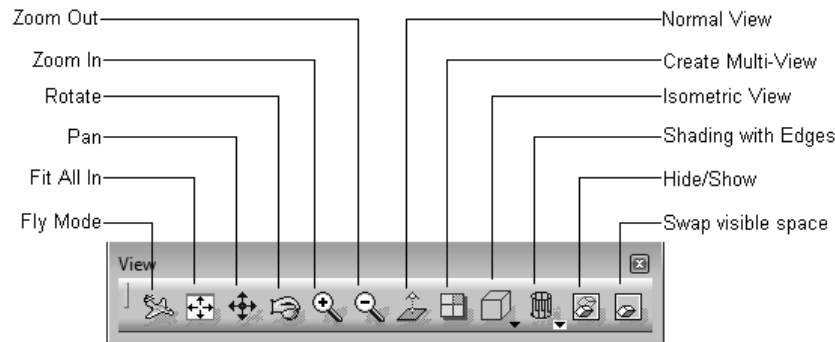


Figure 1-14 The **View** toolbar

Select Toolbar

The **Select** tool is invoked from the **Select** toolbar to select a particular object or sketch. When you invoke the **Select** tool, it prompts you to select an object or a tool. By default, the **Select** tool remains active, until another tool or object is selected. Figure 1-15 shows the **Select** toolbar.

Sketcher Toolbar

The **Sketcher** button in the **Sketcher** toolbar is used to invoke the **Sketcher** workbench. You can also invoke it from the main menu bar by choosing **Start > Mechanical Design > Sketcher**. Figure 1-16 shows the **Sketcher** toolbar.

After choosing the **Sketcher** button, select a plane, or a planar face to invoke the **Sketcher** workbench. The toolbars in the **Sketcher** workbench are discussed next.



Figure 1-15 The **Select** toolbar



Figure 1-16 The **Sketcher** toolbar

Profile Toolbar

The tools in the **Profile** toolbar are used to draw the sketches. It is one of the most important toolbars in the **Sketcher** workbench. Figure 1-17 shows the buttons that are available in this toolbar.

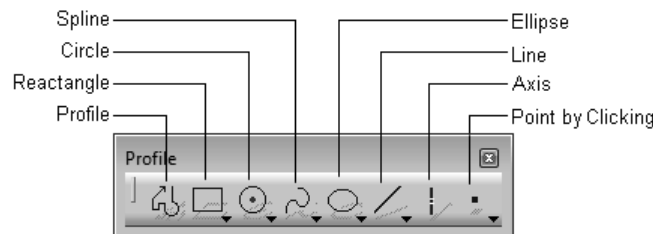


Figure 1-17 The **Profile** toolbar

Constraint Toolbar

The tools in the **Constraint** toolbar are used to apply constraints to the geometric entities, and assign dimensions to a drawn sketch. You can make a sketch fully defined by using the tools from this toolbar. A fully defined sketch is known as an Iso-constraint sketch and is discussed in the later chapters. Figure 1-18 shows the buttons in the **Constraint** toolbar.

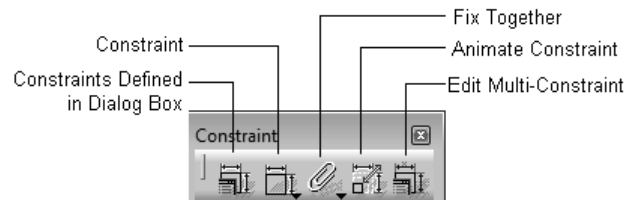


Figure 1-18 The **Constraint** toolbar

Operation Toolbar

The tools in the **Operation** toolbar are used to edit the drawn sketches. Figure 1-19 shows the buttons in the **Operation** toolbar.

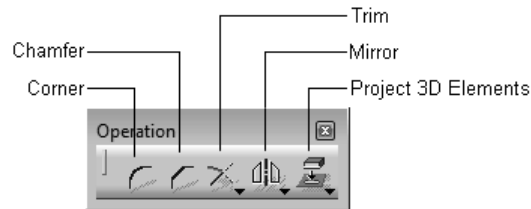


Figure 1-19 The **Operation** toolbar

Sketch tools Toolbar

The tools in the **Sketch tools** toolbar are used to set the sketcher settings such as setting the snap, switching between the standard and construction elements, and so on. Figure 1-20 shows the buttons in the **Sketch tools** toolbar.

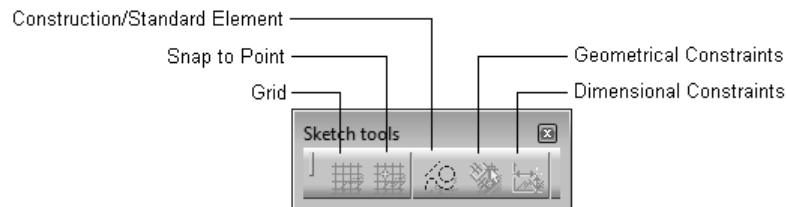


Figure 1-20 The **Sketch tools** toolbar

Once the basic sketch is complete, you need to convert it into a feature. Choose the **Exit workbench** button from the **Workbench** toolbar and switch back to the **Part Design** workbench.

The remaining toolbars of the **Part** workbench are discussed next.

Sketch-Based Features Toolbar

The tools in this toolbar are used to convert a sketch drawn in the **Sketcher** workbench into a feature. Figure 1-21 shows the buttons in the **Sketch-Based Features** toolbar.

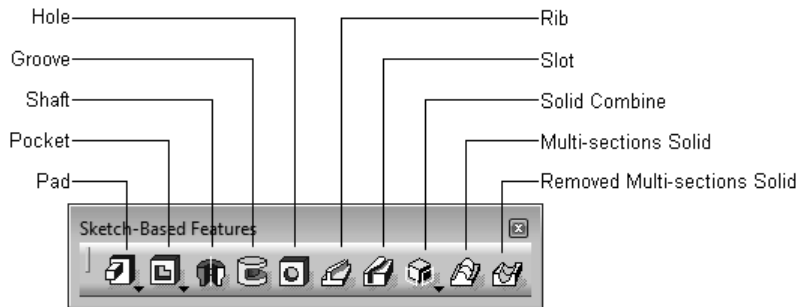


Figure 1-21 The Sketch-Based Features toolbar

Dress-Up Features Toolbar

The tools in the **Dress-Up Features** toolbar are used to apply the dress-up features such as fillet, chamfer, shell, and so on. Figure 1-22 shows the buttons in the **Dress-Up Features** toolbar.

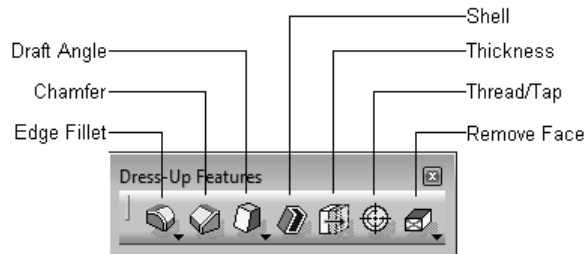


Figure 1-22 The Dress-Up Features toolbar

Measure Toolbar

The tools in the **Measure** toolbar are used to measure a single item, measure the distance between two geometries, or calculate the mass properties of the object. Figure 1-23 shows the buttons in the **Measure** toolbar.

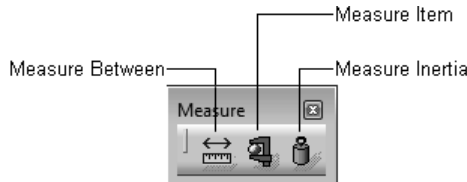


Figure 1-23 The Measure toolbar

Transformation Features Toolbar

The tools in the **Transformation Features** toolbar are used to apply the transformation features to the parts such as moving, mirror, pattern, and so on. Figure 1-24 shows the buttons in the **Transformation Features** toolbar.

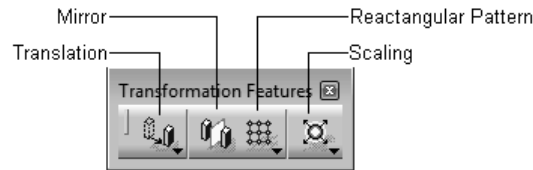


Figure 1-24 The Transformation Features toolbar

Dynamic Sectioning Toolbar

The tool in the **Dynamic Sectioning** toolbar is used to visualize the 3D section view of a part at a position specified by using a section plane. Figure 1-25 shows the **Dynamic Sectioning** toolbar.



Figure 1-25 The Dynamic Sectioning toolbar



Figure 1-26 The Apply Material toolbar

Apply Material Toolbar

The tool in the **Apply Material** toolbar is used to assign a material to the part body. Figure 1-26 shows the **Apply Material** toolbar.

Surface-Based Features Toolbar

The tools in the **Surface-Based Features** toolbar are used to perform surface-based operations on part bodies or to convert a surface body into a solid body. Figure 1-27 shows the buttons in the **Surface-Based Features** toolbar.

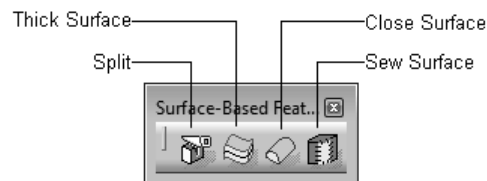


Figure 1-27 The Surface-Based Features toolbar

Wireframe and Surface Design Workbench Toolbars

You can invoke the **Wireframe and Surface Design** workbench from the main menu bar by choosing **Start > Mechanical Design > Wireframe and Surface Design**. The toolbars in the **Wireframe and Surface Design** workbench are discussed next.

Surfaces Toolbar

The tools in the **Surfaces** toolbar are used to create surfaces. Figure 1-28 shows the **Surfaces**

toolbar.

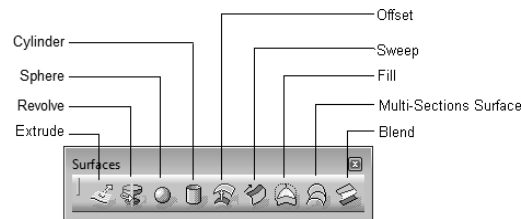


Figure 1-28 The Surfaces toolbar

Operations Toolbar

The tools in the **Operations** toolbar are used for surface editing operations. Figure 1-29 shows the buttons in the **Operations** toolbar.

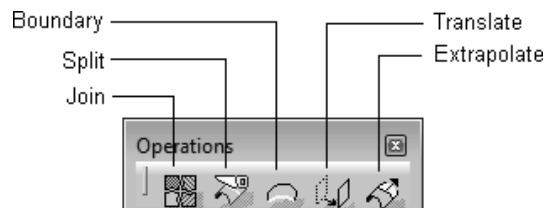


Figure 1-29 The Operations toolbar

Wireframe Toolbar

The tools in the **Wireframe** toolbar are used to create the 2D or 3D curve using points, lines, and splines. User-defined planes can also be created by choosing the **Plane** button from this toolbar. You will learn about creating a user-defined plane in the later chapters. Figure 1-30 shows the buttons in the **Wireframe** toolbar.

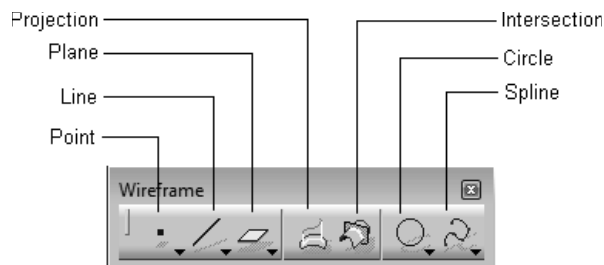


Figure 1-30 The Wireframe toolbar

Assembly Design Workbench Toolbars

You can invoke the **Assembly Design** workbench by choosing the **New** button from the **Standard** toolbar and then selecting **Product** from the **New** dialog box. Alternatively, you can choose **Start > Mechanical Design > Assembly Design** from the menu bar. The toolbars in the **Assembly Design** workbench are discussed next.

Product Structure Tools Toolbar

The tools in the **Product Structure Tools** toolbar are used to insert an existing part or

assembly in the current product file. You can also create a new assembly or part inside the Product file using the tools in this toolbar. Figure 1-31 shows the buttons in the **Product Structure Tools** toolbar.

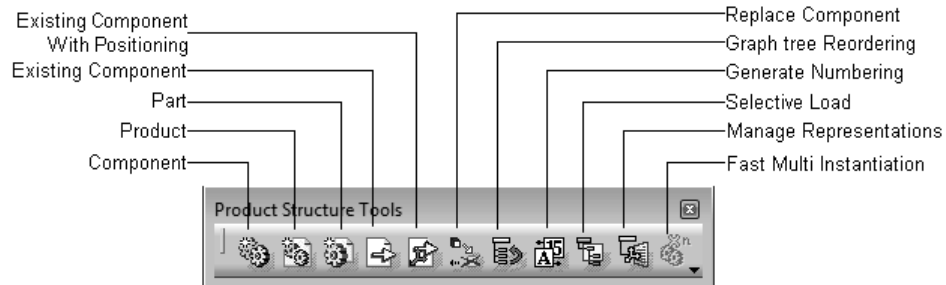


Figure 1-31 The Product Structure Tools toolbar

Constraints Toolbar

The **Constraints** toolbar is used to apply constraints to the components of the assembly to restrict degrees of freedom (DOFs) of the component with respect to the surroundings. Figure 1-32 shows the **Constraints** toolbar.

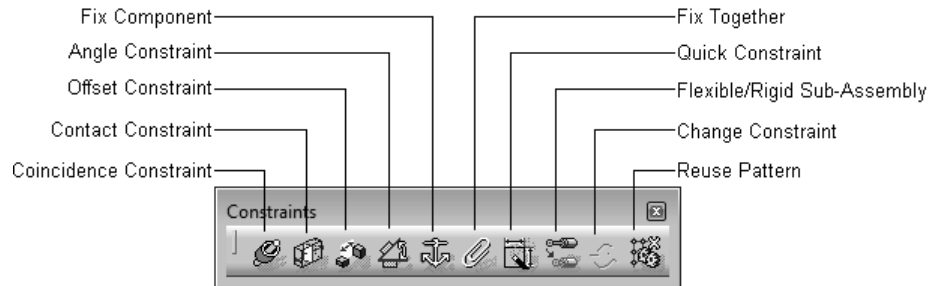


Figure 1-32 The Constraints toolbar

Move Toolbar

The tools in this toolbar are used to perform tasks such as moving and snapping the parts or exploding an assembly in the **Assembly Design** workbench. Figure 1-33 shows the **Move**

toolbar.

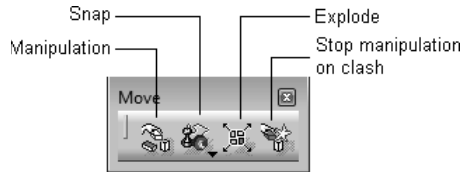


Figure 1-33 The Move toolbar

Space Analysis Toolbar

The tools in the **Space Analysis** toolbar are used to check any interference and clash in the assembly, create a section of the assembly, and perform the distance analysis. Figure 1-34 shows the buttons in the **Space Analysis** toolbar.

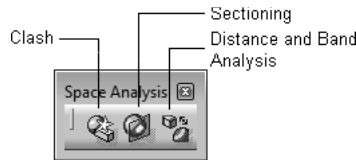


Figure 1-34 The Space Analysis toolbar

Drafting Workbench Toolbars

To invoke the **Drafting** workbench, choose the **New** button from the **Standard** toolbar and select **Drawing** option from the **New** dialog box. Choose the **OK** button in the **New Drawing** dialog box. Alternatively, choose **Start > Mechanical Design > Drafting** from the menu bar to invoke this workbench. The toolbars in **Drafting** workbench are discussed next.

Drawing Toolbar

The tools in the **Drawing** toolbar are used to insert a new sheet, create a new view, and so on. Figure 1-35 shows the **Drawing** toolbar.

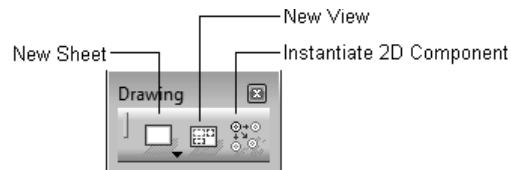


Figure 1-35 The Drawing toolbar

Views Toolbar

The tools in the **Views** toolbar are used to generate orthographic, section, detail, or clipped view for a solid part or assembly. Figure 1-36 shows the **Views** toolbar.

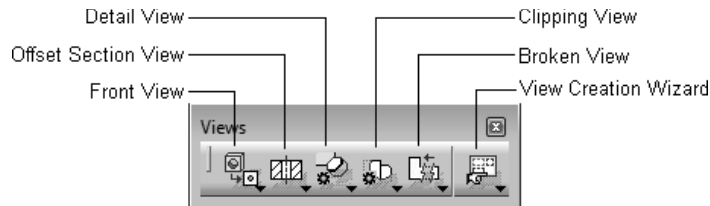


Figure 1-36 The Views toolbar

Generation Toolbar

The tools in the **Generation** toolbar are used to generate dimensions and assign balloons to the assembly drawings. Figure 1-37 shows the **Generation** toolbar.



Figure 1-37 The Generation toolbar

Other toolbars in the **Drafting** workbench are similar to those discussed in the **Sketcher** workbench. The tools in these toolbars are discussed in the later chapters.

You will notice a down arrow at the bottom right corner of the buttons in most of the toolbars. When you choose this arrow, a flyout will appear. Figure 1-38 shows the flyout that appears when the down arrow of the **Point** button is chosen from the **Geometry Creation** toolbar.



Note

The flyout, which appears on choosing the down arrow, becomes an independent toolbar if you click on the line available on its top, refer to Figure 1-38. This line can be on the top or left of the flyout, depending on whether the flyout is horizontal or vertical.

Generative Sheetmetal Toolbars

This workbench can be invoked by choosing **Start > Mechanical Design > Generative Sheetmetal Design** from the menu bar. The toolbars in the **Generative Sheetmetal Design** are discussed next.

Walls Toolbar

The tools in the **Walls** toolbar are used to create different types of walls. Figure 1-39 shows the **Walls** toolbar.

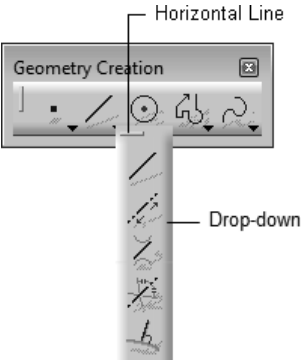


Figure 1-38 The drop-down that appears when a down arrow is chosen

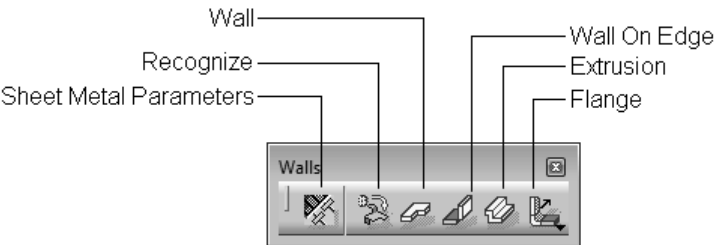


Figure 1-39 The **Walls** toolbar

Swept Walls Toolbar

The tools in the **Swept Walls** toolbar are used to create sweep features. These features are created by sweeping a profile along the spline. Figure 1-40 shows the **Swept Walls** toolbar.

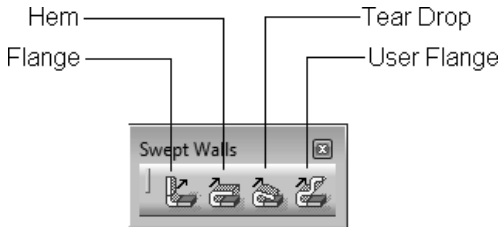


Figure 1-40 The **Swept Walls** toolbar

Stamping Toolbar

The tools in the **Stamping** toolbar are used to create different types of stamping features. Figure 1-41 shows the **Stamping** toolbar.

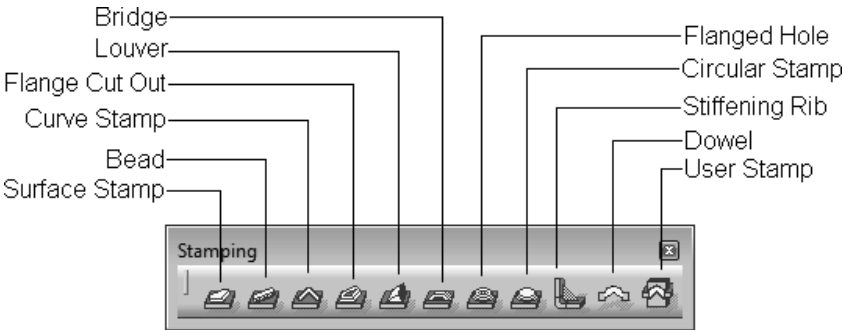


Figure 1-41 The **Stamping** toolbar

DMU Kinematics Toolbars

This **DMU Kinematics** workbench can be invoked by choosing **Start > Digital Mockup > DMU Kinematics** from the menu bar. The toolbars in the **DMU Kinematics** environment are discussed next.

DMU Kinematics Toolbar

The tools in the **DMU Kinematics** toolbar are used to create mechanisms and simulate the mechanism. Figure 1-42 shows the **DMU Kinematics** toolbar.

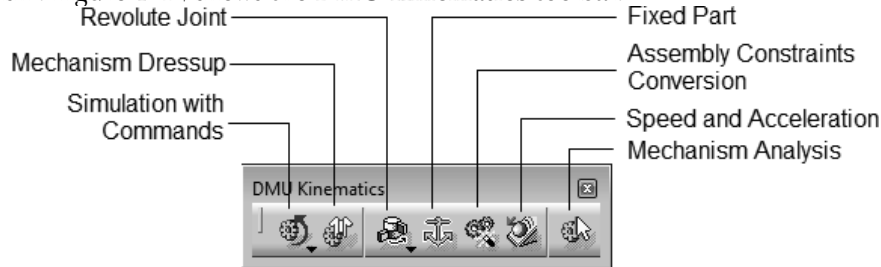


Figure 1-42 The *DMU Kinematics* toolbar

Kinematics Joints Toolbar

The tools in the **Kinematic Joints** toolbar are used to create different type of joints in the mechanism. Figure 1-43 shows the **Kinematic Joints** toolbar.

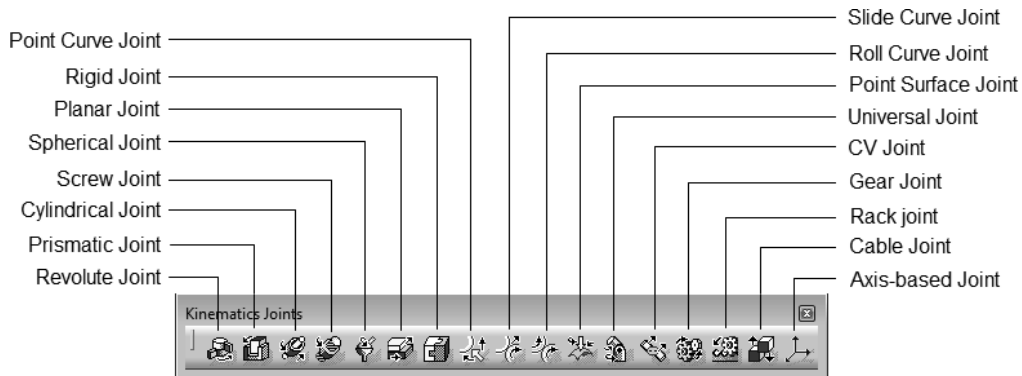


Figure 1-43 The *Kinematics Joints* toolbar

HOT KEYS

CATIA V5 is more popularly known for its icon driven structure. However, you can still use the keys on the keyboard to invoke some tools. These keys are called hot keys. The hot keys along with their functions are listed in the table given next.

Hot Key	Function
CTRL+Z	Invokes the Undo tool
CTRL+Y	Invokes the Repeat tool

CTRL+S	Saves the current document
ALT+ENTER	Invokes the Properties tool
CTRL+F	Invokes the Search tool
CTRL+U	Invokes the Update tool
SHIFT+F2	Invokes the Specification Overview tool
F3	Toggles the display of the specification tree
SHIFT+F1	Invokes the What's This? tool
F1	Invokes the CATIA V5 Help tool
CTRL+D	Invokes the Fast Multi-Instantiation tool in the Assembly Design workbench

COLOR SCHEME

CATIA V5 allows you to use various color schemes as the background screen color, and also for displaying the entities on the screen. To change the color scheme, choose **Tools > Options** from the menu bar; the **Options** dialog box will be displayed. Choose **General > Display** option from the left pane of the **Options** dialog box; the tabs corresponding to this selection will be displayed on the right in the **Options** dialog box. Choose the **Visualization** tab and select the **White** color from the **Background** drop-down list. Choose **OK** to apply the scheme to the CATIA V5 environment. Note that all files that you open henceforth will use this color scheme. The default color of the sketched elements is white, therefore, you need to change the colors of the sketch elements to black. To change the default color, invoke the **Options** dialog box and expand the **Mechanical Design** branch from the left of this dialog box. Next, expand the **Sketcher** branch and choose **Black** from the **Default color of the elements** drop-down list. Choose the **OK** button from the **Options** dialog box. All sketches drawn after setting this option will be displayed in black.

To distinguish between a plane and a background, you need to change the default color of the planes. To do so, select the plane and right-click to invoke the shortcut menu. Select the **Properties** option from it; the **Properties** dialog box is displayed. Choose the **Graphic** tab and then select the **Black** color from the drop-down list in the **Lines and Curves** area. Choose the **OK** button to apply the scheme to the CATIA V5 environment. Similarly, change the color of the remaining two planes. Figure 1-44 shows the **Graphic** tab of the **Properties** dialog box.

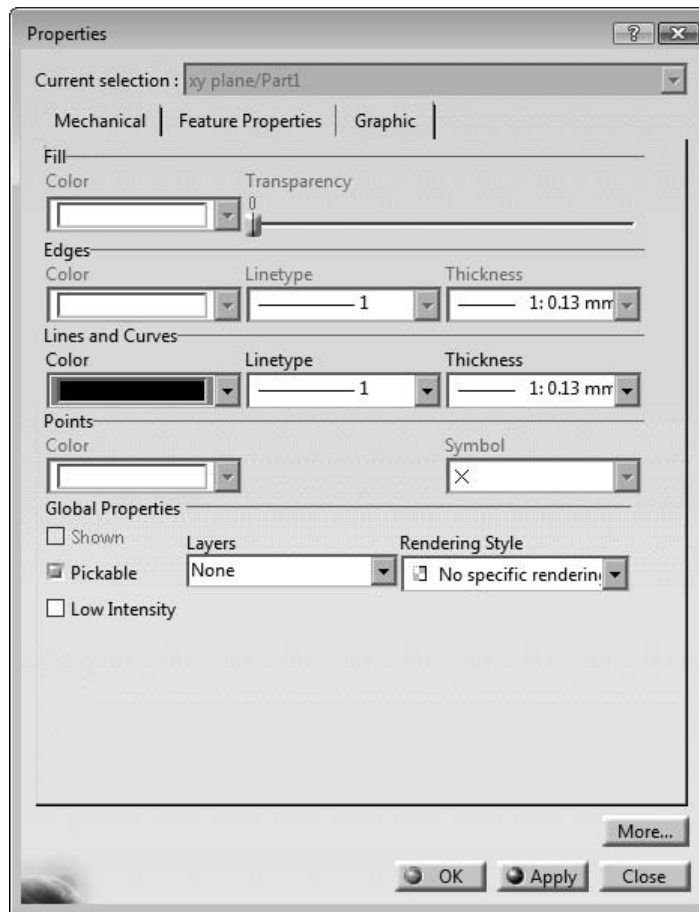


Figure 1-44 The **Properties** dialog box



Note

On changing the background color of the screen to White, you will have to change the color of all planes, every time you open a new file. The planes include the default xy, yz, and zx planes and also the planes that will be created during the development of the design. As it is very cumbersome to perform this activity every time, it is recommended that you do not change the background color of the CATIA V5 environment.

For the purpose of printing, this book will follow the white background of the CATIA V5 environment. However, for better understanding and clear visualization at various places, this book will follow other color schemes too.