



طراحی به کمک رایانه

فصل ۱: مقدمه

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CATIA V5R16

طراحی به کمک رایانه

ساخت به کمک رایانه

- Welcome to **CATIA** (**Computer Aided Three Dimensional Interactive Application**).
- As a new user of this software package, you will be joining 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 state of the art user interface, CATIA V5 delivers innovative technologies, for maximum productivity and creativity from the concept to the final product.
- CATIA V5 reduces the learning curve for the user, as it allows the flexibility of using | feature based and parametric designs.

مهندسی به کمک رایانه

- 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, which allow the user to perform the specific design tasks in a particular area.
- The basic workbenches available in CATIA V5 are : محیط های اصلی نرم افزار

- **Part Design Workbench** محیط طراحی سه بعدی

The **Part Design** workbench is a parametric and feature-based environment, in which you can create solid models.

- **Wireframe and Surface Design Workbench** محیط طراحی سیمی و سطحی

- The **Wireframe and Surface Design** workbench is also a parametric and feature-based environment, in which you can 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** محیط مونتاژ
 - 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:
 - Bottom-up
 - Top-down
- **Drafting Workbench** محیط نقشه کشی
 - The **Drafting** workbench is used for the documentation of the parts or the assemblies created earlier in the form of drawing views and their detailing.
 - There are two types of drafting techniques:
 - Generative drafting
 - Interactive drafting

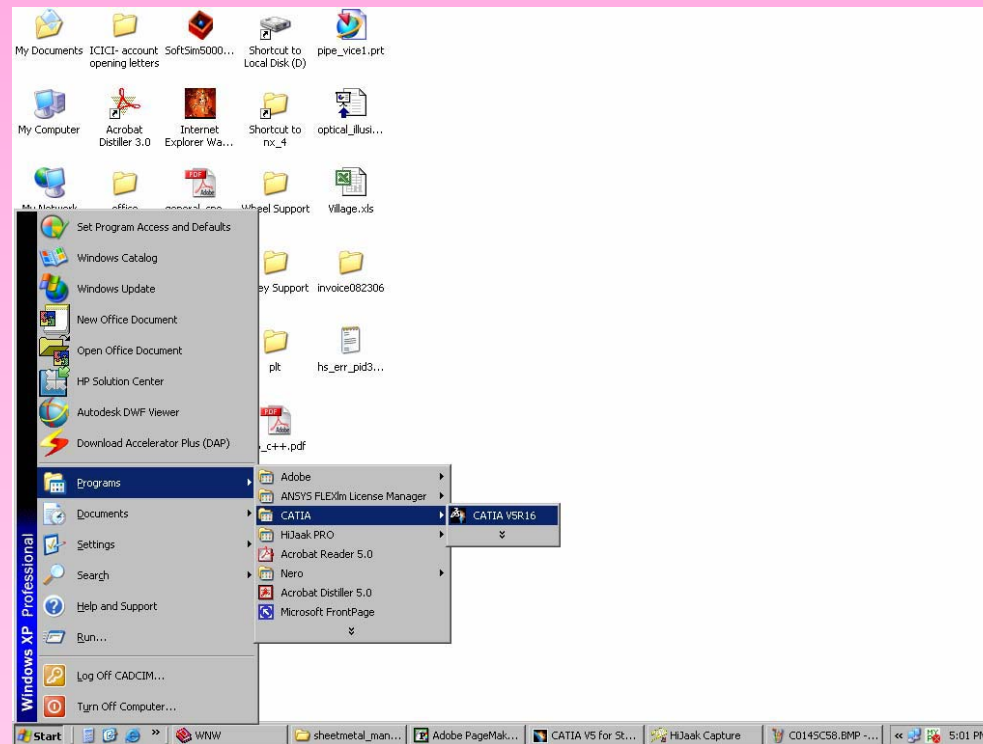
➤ **SYSTEM REQUIREMENTS** سیستم مورد نیاز برای نصب برنامه

The following are the system requirements to ensure smooth running of CATIA V5R16 on your system:

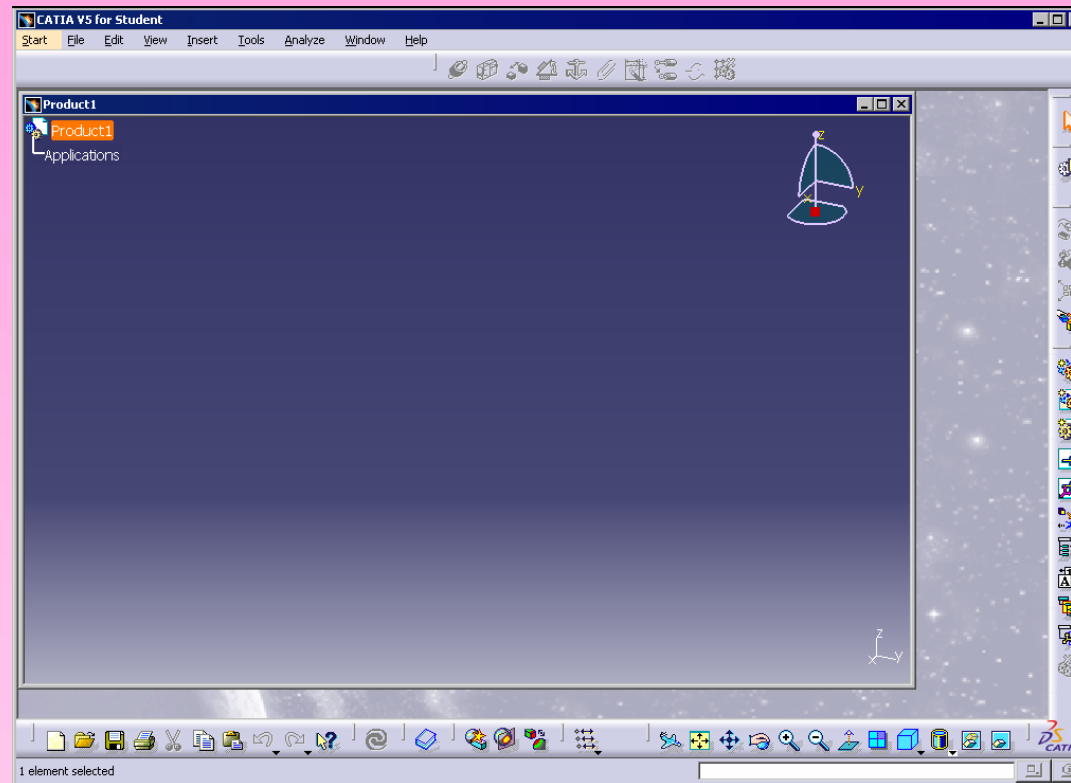
- System unit: An Intel Pentium III or Pentium 4 based workstation running Microsoft 2000 Professional Edition or Windows XP Professional Edition.
- Memory: 256 MB of RAM is the minimum recommended for all applications. 512 MB of RAM is recommended for DMU applications.
- Disk drive: 4 GB Disk Drive space (Minimum recommended size)
- Internal/External drives: A CD-ROM drive is required for program installation.
- 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 is required with minimum resolution of 1024x768 for Microsoft Windows workstations and 1280x1024 for UNIX workstations.

➤ GETTING STARTED WITH CATIA V5R16 شروع کار با کتیا

- Install CATIA V5R16 on your system and then start it by double-clicking on the shortcut icon of **CATIA V5R16** on the desktop of your computer.
- You can also choose **Start > Programs > CATIA > CATIA V5R16** from the taskbar menu, as shown in the figure.



Starting **CATIA V5R16** using the taskbar shortcuts



The initial screen that appears after starting CATIA V5R16

محیط کلی اولیه نرم افزار

محیط کاربری کلی نرم افزار کتیا



The screen that appears after closing the initial product file

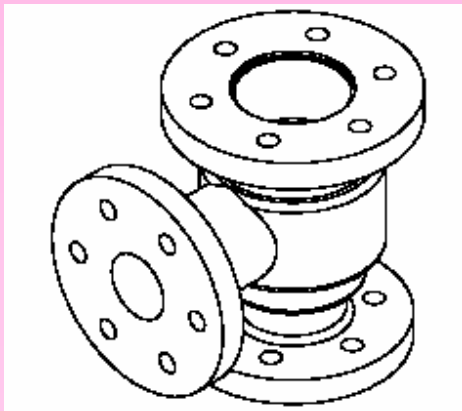
➤ IMPORTANT TERMS AND DEFINITIONS اجزا و تعاریف مهم نرم افزار

• Feature-based Modeling مدلسازی بر اساس ویژگی

- 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 individual features and each feature is related to the other directly or indirectly.

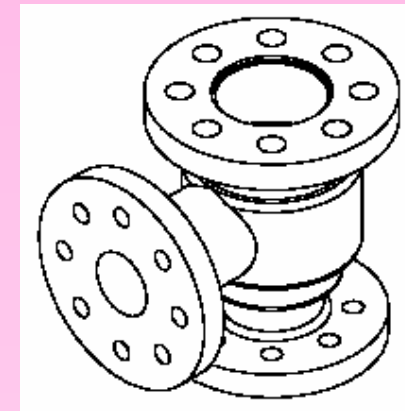
• Parametric Modeling مدلسازی پارامتریک

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.



Body of a pipe housing

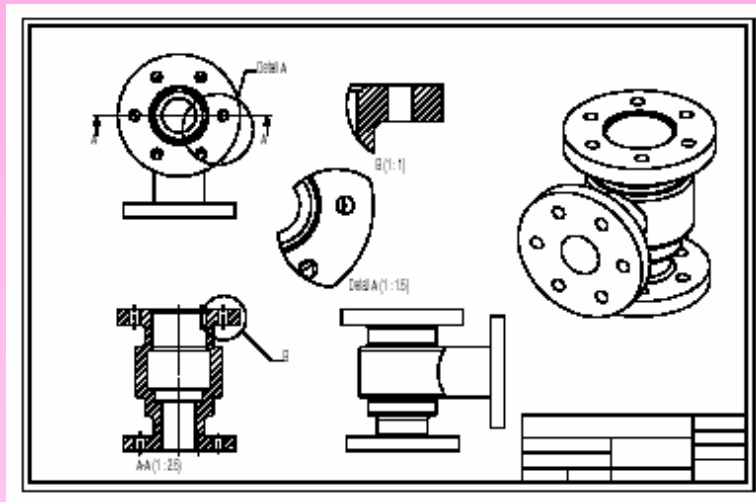
مثال: چرخ دنده و اتصالات



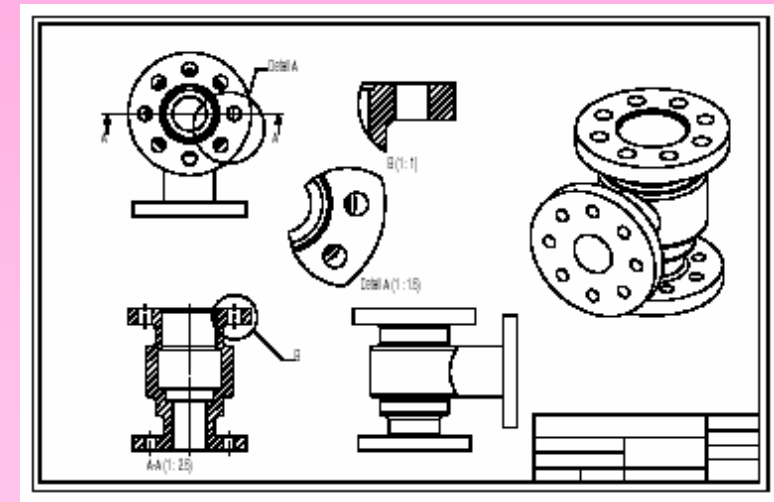
Modified body of pipe housing

- **Bidirectional Associativity** *تعامل دوسویه*

The bidirectional associativity ensures that if any modification is made in the model in any one of the workbenches of CATIA V5, it is automatically reflected in the other workbenches immediately.



The drawing views of the body part before making the modifications



The drawing views, after modifications

تعامل دوسویه محیط های طراحی مختلف

- **CATPart** پسوند فایل های ذخیره شده محیط طراحی سه بعدی

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

- **CATProduct** محیط مونتاژ

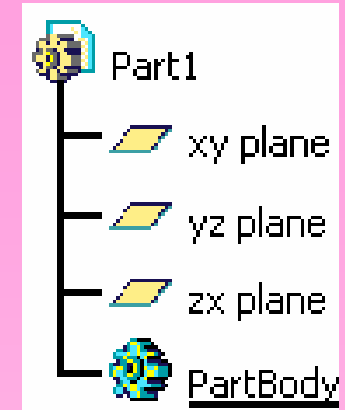
CATProduct is a file extension associated with all the files that are created in **Assembly Design** workbench of CATIA V5.

- **CATDrawing** محیط نقشه برداری

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

• Specification Tree *درخت طراحی*

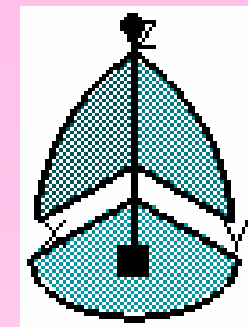
- The specification tree keeps a track of all the operations that are carried on the part, as shown in the figure.
- The specification tree that appears when you start a new file under the **Part Design** workbench, is as shown in the figure.



The specification tree that appears on starting a new CATPart file

• Compass *قطب نما*

- It is a tool that is used to manipulate the orientation of parts, assemblies, or sketches.
- You can also orient the view of the parts and assemblies.
- By default, it appears on the top right corner of the geometry area.



The Compass

- **Constraints** قید/اندازه گذاری
 - Constraints are logical operations that are performed on the selected element to define its size and location with respect to other elements or reference geometries.
 - The constraints in **Sketcher** workbench are called geometric constraints and the constraints available in the **Assembly Design** workbench are called assembly constraints.
- **Geometric Constraints** قید هندسی

These are the logical operations performed on sketched elements to define their size and position with respect to other elements.

The constraints available in the **Sketcher** workbench are:

- **Distance** فاصله
- **Length** طول
- **Angle** زاویه
- **Radius / Diameter** شعاع/قطر
- **Semimajor axis**
- **Semiminor axis**
- **Symmetry** تقارن
- **Midpoint** نقطه میانی
- **Equidistant point** نقاط هم فاصله
- **Fix** ثابت
- **Coincident** منطبق
- **Concentricity** هم مرکزی
- **Tangency** مماسی
- **Parallelism** موازی
- **Perpendicular** تعامد
- **Horizontal** افقی
- **Vertical** عمودی

- **Assembly Constraints** قیدهای محیط مونتاژ

Constraints available in the **Assembly Design** workbench are logical operations performed to restrict the degree of freedom of the component and to precisely define their location and position with respect to other components of the assembly.

- **Coincidence Constraint** تطابق

- **Fix Component** ثابت

- **Contact Constraint** تماس

- **Fix Together** ثابت نسبت به یکدیگر

- **Offset Constraint** فاصله

- **Quick Constraint** قیدگذاری سریع

- **Angle Constraint** زاویه

- **PartBody**

- It is the default body available under **Part Design** workbench.
- All the solid related features, such as pad, pocket, shaft, and so on are placed inside it.

- **Geometrical Set** مجموعه اجزای هندسی

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

- **Wireframe** اجزای سیمی

The wireframe construction elements aid in creating surfaces and are used as a substitute to entities drawn in the **Sketcher** workbench.

- **Surface** سطوح

- Surface are geometric feature which have no thickness.
- They are generally used to create complex shapes that are difficult to create using the solid feature.

- **Feature** ویژگی های هندسی

- A features is defined as a basic building block of a solid model.
- The combination of various features results in a complete model.

- **Reframe on** *حالت نمایش تمام اجزا*

Sometimes a feature, body, or a sketch may not be visible in the available space in the geometry area, this operation is used to view that particular selection in the available display space.

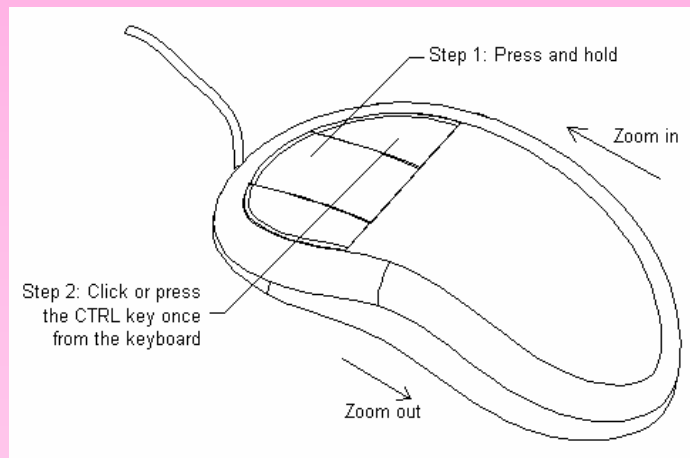
- **Center Graph**

This option brings 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 *کار با ماوس و کلیدهای ترکیبی*

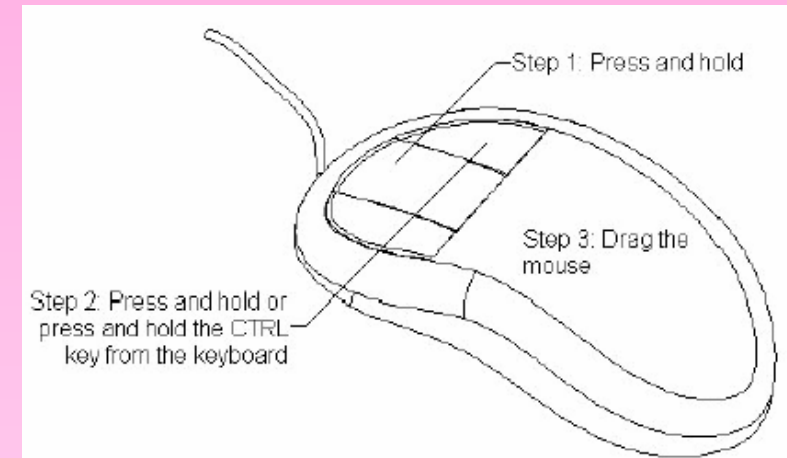
- To work with CATIA V5 design workbenches, it is necessary that you understand the functions of the mouse buttons.
- The efficient usage of these three buttons, along with the CTRL key on the keyboard, can reduce the time involved in completing the design task.

اعمال بزرگنمایی و کوچک نمایی



Using the three button mouse to perform the zoom in and zoom out operations

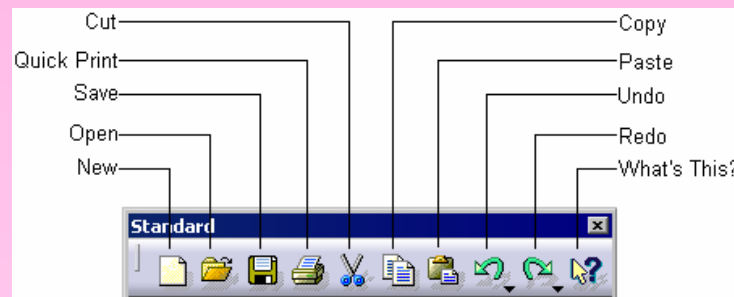
اعمال چرخش



Using the three button mouse to perform the rotate operation

➤ TOOLBARS *نوار ابزار*

- CATIA V5 offers a user-friendly design environment by providing specific toolbars to each workbench.
- The toolbars that appear in various workbenches are:
- **Standard Toolbar** *نوار ابزار استاندارد*
 - This toolbar is common to all workbenches of CATIA V5.
 - The following figure shows the Standard toolbar.



The Standard toolbar

نوار ابزار استاندارد

- **Status Bar** نوار وضعیت

The **Status** bar, which appears at the bottom of the CATIA V5 window, comprises of three areas, as shown in the figure.

اطلاعات کنونی یا کادر محاوره

دکمه نمایش کادر محاوره



The status Bar نوار وضعیت

- **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.

- **Dialog Box Display Button**

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

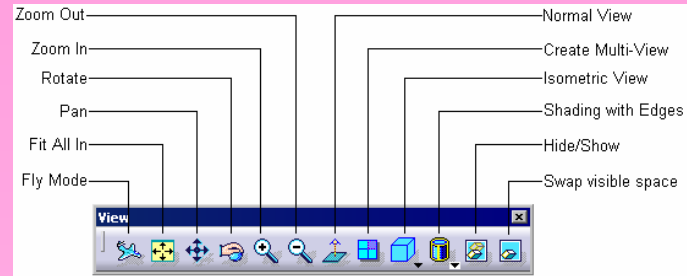
- **User Information Package Button**

Choosing this button will open a window with a default link that is C:\Program Files\Dassault Systemes\B16\intel_a\resources\galaxy\default.htm.

- **Part Design Workbench Toolbars** *نوار ابزارهای محیط دوبعدی*

You can invoke the **Part Design** workbench by choosing the **New** button from the **Standard** toolbar and selecting **Part** from the **New** dialog box.

- View Toolbar



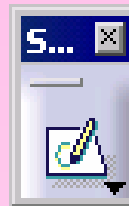
The View toolbar نوار ابزار نمایش

- Select Toolbar



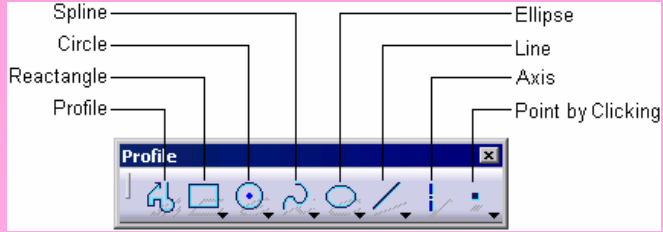
The Select toolbar نوار ابزار انتخاب

- Sketcher Toolbar



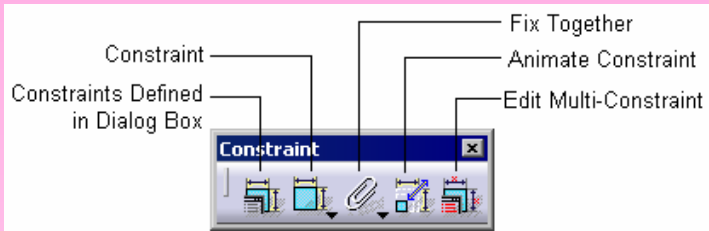
The Sketcher toolbar نوار ابزار محیط اسکچ

- Profile Toolbar



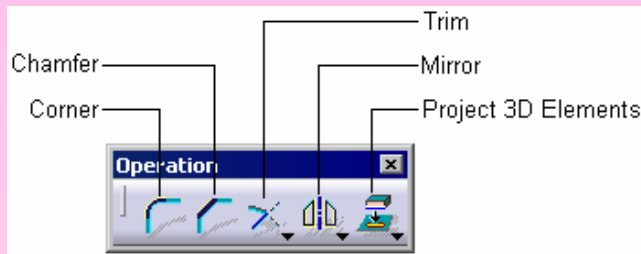
The Profile toolbar نوار ابزار ترسیم های دوبعدی

- Constraint Toolbar



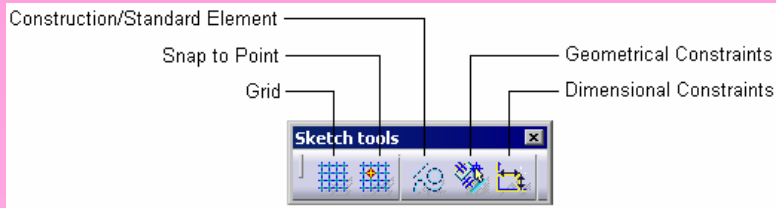
The Constraint toolbar نوار ابزار قید

- Operation Toolbar



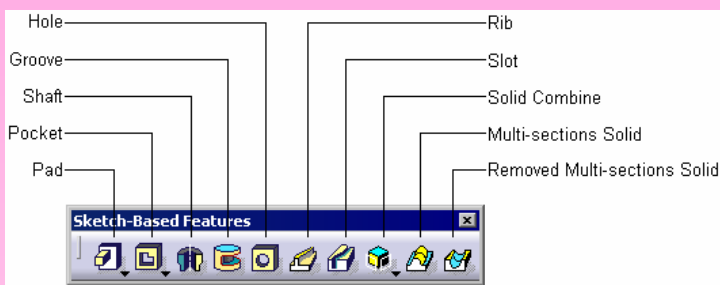
The Operation toolbar نوار ابزار عملیات ویرایش

- Sketch tools Toolbar



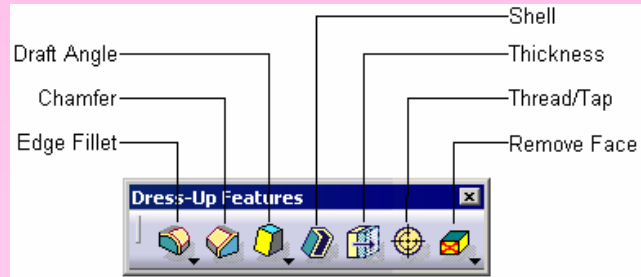
The Sketch tools toolbar *نوار ابزار تنظیمات اسکچ*

- Sketch-Based Features Toolbar



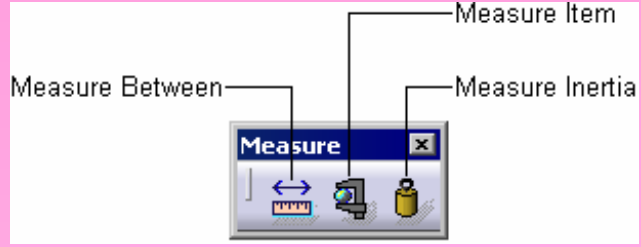
The Sketch-Based Features toolbar *نوار ابزار تعریف حجم*

- Dress-Up Features Toolbar



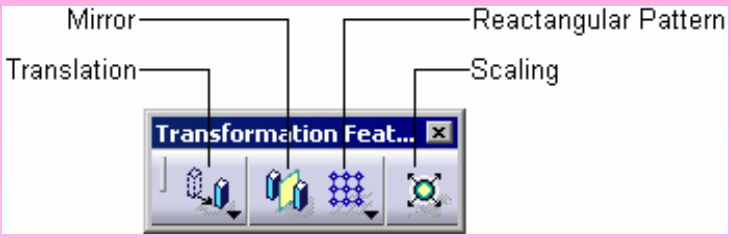
The Dress-Up Features toolbar *نوار ابزار ویرایش قطعه*

• Measure Toolbar



The Measure toolbar نوار ابزار اندازه گیری

• Transformation Features Toolbar



The Transformation Features toolbar نوار ابزار جابجایی

• Surface-Based Features Toolbar



The Surface-Based Features toolbar نوار ابزار تعریف سطح

- **Apply Material Toolbar**

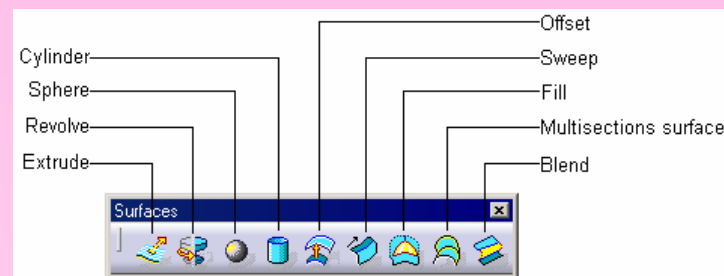


The **Apply Material** نوار ابزار اعمال ماده toolbar

- **Wireframe and Surface Design Workbench Toolbars**

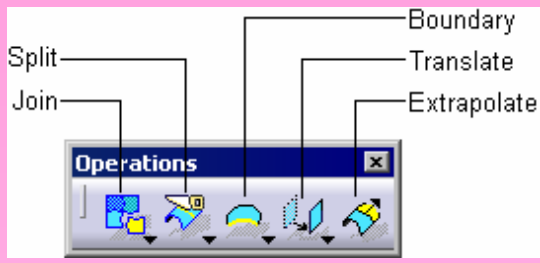
You can invoke the **Wireframe and Surface Design** workbench from the main menu bar by choosing **Start > Mechanical Design > Wireframe and Surface Design**.

- **Surfaces Toolbar**



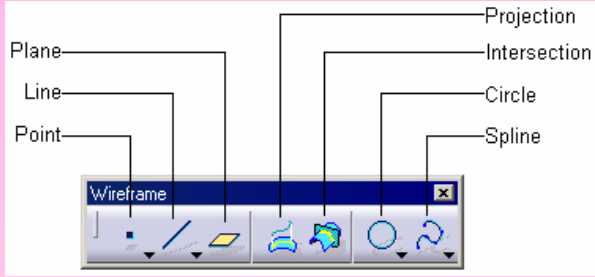
The **Surfaces toolbar** نوار ابزار ایجاد سطح

- Operations Toolbar



The Operations toolbar *نوار ابزار ویرایش سطح*

- Wireframe Toolbar

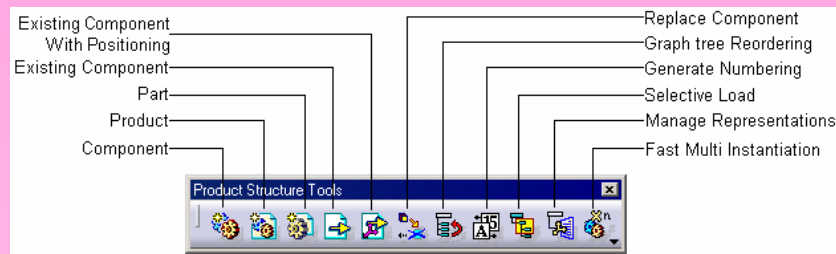


The Wireframe toolbar *نوار ابزار ایجاد المان های سیمی*

- **Assembly Design Workbench Toolbars**

You can invoke the **Assembly Design** workbench by choosing the **New** button from the **Standard** toolbar and selecting **Product** from the **New** dialog box.

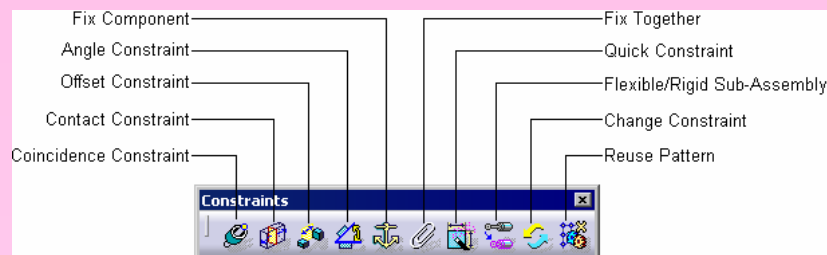
- **Product Structure Tools Toolbar**



The Product Structure Tools toolbar

نوار ابزار تعریف مجموعه مونتاژ

- **Constraints Toolbar**



The Constraints toolbar

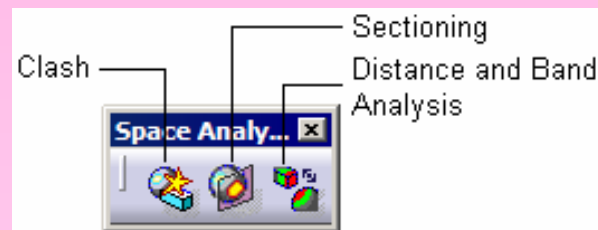
نوار ابزار قید گذاری

- **Move Toolbar**



The Move toolbar نوار ابزار جابجایی

- **Space Analysis Toolbar**

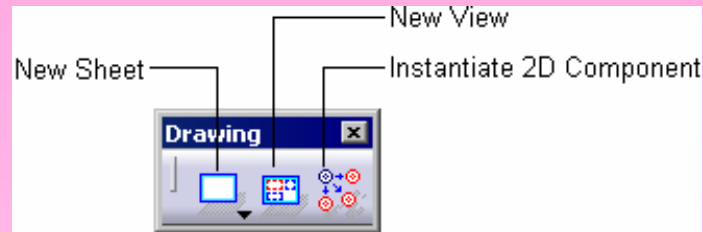


The Space Analysis toolbar نوار ابزار انالیز حجم

- **Drafting Workbench Toolbars**

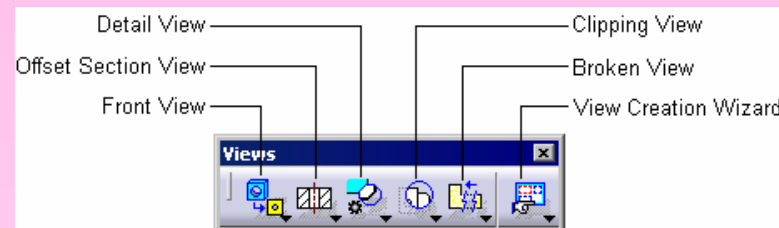
To invoke the **Drafting** workbench, choose the **New** button from the **Standard** toolbar and select **Drawing** from the **New** dialog box.

- **Drawing Toolbar**



The **Drawing toolbar** نوار ابزار ترسیم

- **Views Toolbar**



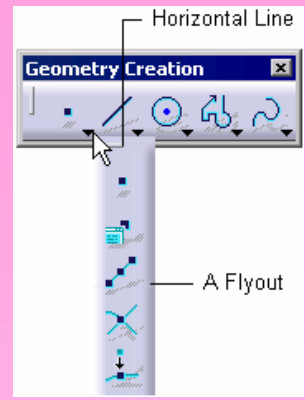
The **Views toolbar** نوار ابزار نمایش

• Generation Toolbar *اضافه کردن اجزای هندسی*



نوار ابزار ایجاد نما

The **Generation toolbar**



The flyout that appears when a down arrow is chosen

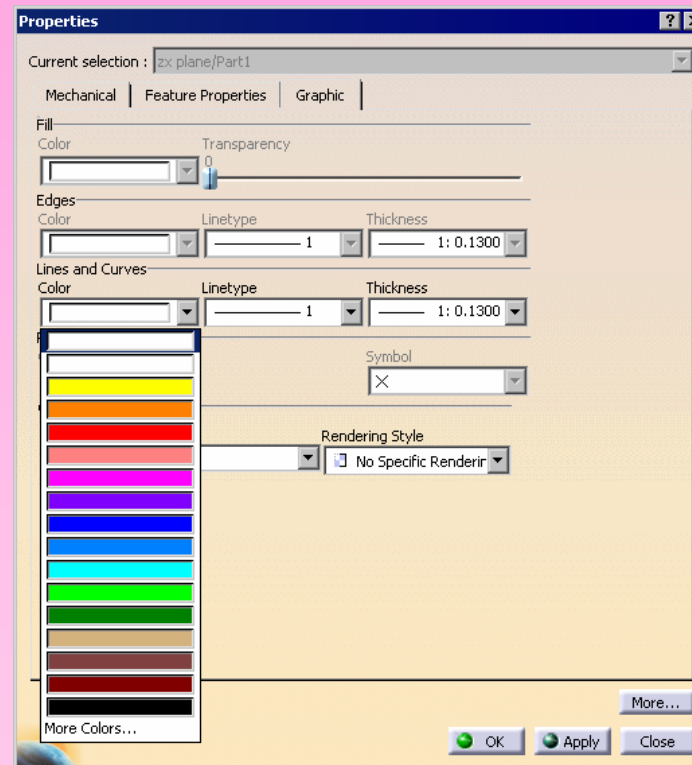
➤ **HOT KEYS** *کلیدهای ترکیبی*

- CATIA V5 is more popularly known for its icon driven structure.
- The hot keys, along with their functions, are listed in the table shown.

Hot Keys	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 MultiInstantiation tool in the Assembly Design workbench
CTRL+E	Invokes the Define MultiInstantiation tool in the Assembly Design workbench

➤ COLOR SCHEME تنظیم رنگ

CATIA allows you to use various color schemes as the background screen color, and also for displaying the entities on the screen.



The Properties dialog box کادر محاوره مشخصات