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# CATIA V5

Dr Ahmed Kovacevic

City University London

School of Engineering and Mathematical Sciences

Room CM124, Phone: 8780, E-Mail: [Design@city.ac.uk](mailto:Design@city.ac.uk)

[www.city-design.tk](http://www.city-design.tk)

[www.staff.city.ac.uk/~ra600/intro.htm](http://www.staff.city.ac.uk/~ra600/intro.htm)

# Course Schedule

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- Lecture 1
  - Introduction
  - Software Overview
  - Part Design and Sketching
- Lecture 2
  - Product Structure and Assembly Modelling
  - More advance Part Design
- Lecture 3
  - Wireframe and Surface
  - Drafting
- Lecture 4
  - Finite Element Analysis
  - Data Exchange
  - Parameters and Formulas

# CATIA Overview

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- CATIA v5 is an Integrated Computer Aided Engineering tool:
  - ❑ Incorporates CAD, CAM, CAE, and other applications
  - ❑ Completely re-written since CATIA v4 and still under development
  - ❑ CATIA v5 is a native Windows application
  - ❑ User friendly icon based graphical user interface
  - ❑ Based on Variational/ Parametric technology
  - ❑ Encourages design flexibility and design reuse
  - ❑ Supports Knowledge Based Design

# CATIA v5 Philosophy

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- A Flexible Modelling environment
  - ❑ Ability to easily modify models, and implement design changes
  - ❑ Support for data sharing, and data reuse
- Knowledge enabled
  - ❑ Capture of design constraints, and design intent as well as final model geometry
  - ❑ Management of non-geometric as well as geometric design information
- The 3D Part is the Master Model
  - ❑ Drawings, Assemblies and Analyses are associative to the 3D parts. If the part design changes, the downstream models with change too.

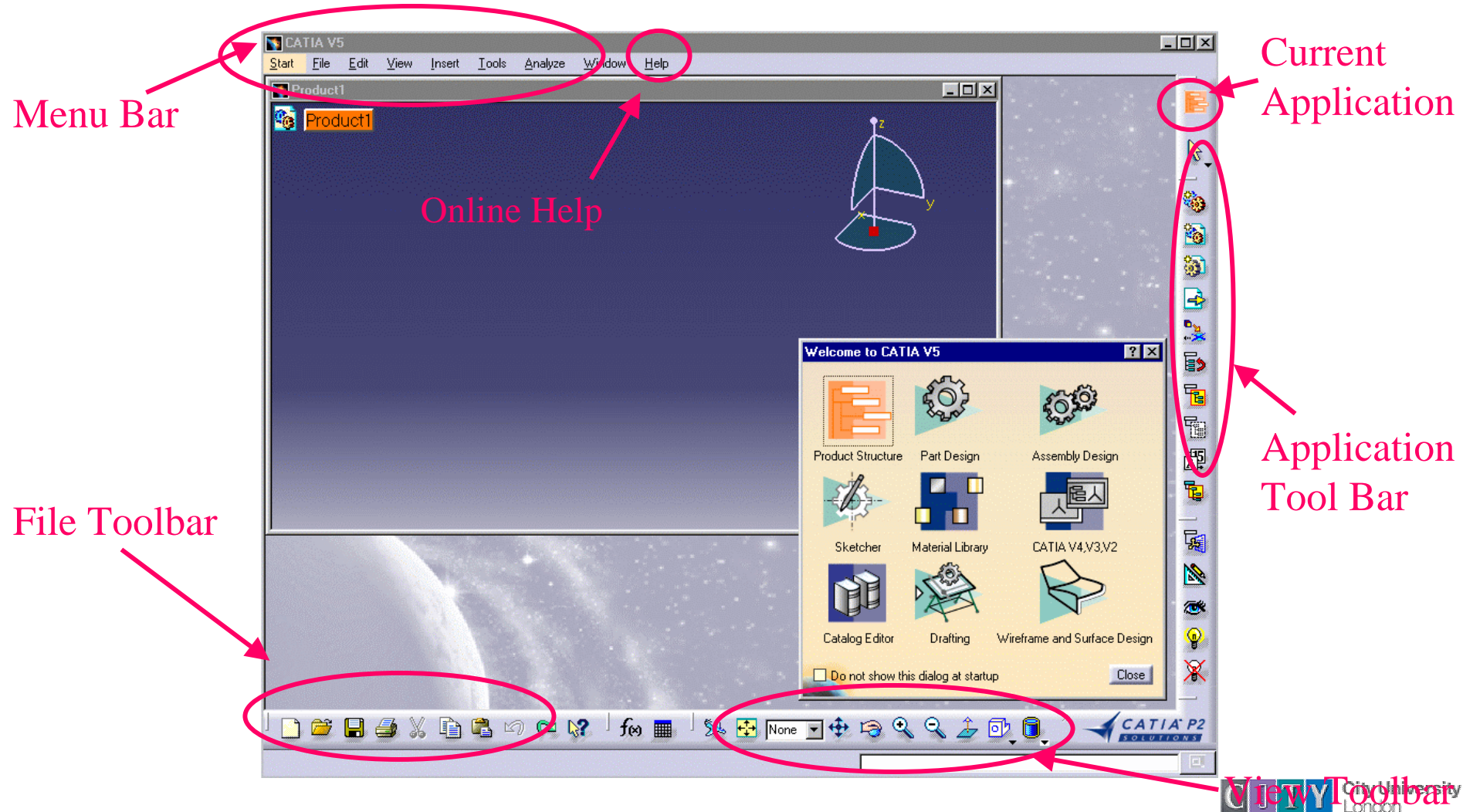
# CATIA v5 Applications

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- Product Structure
- Part Design
- Assembly Design
- Sketcher
- Drafting (Interactive and Generative)
- Wireframe and Surface
- Freestyle Shaper
- Digital Shape Editor
- Knowledgeware
- Photo Studio
- 4D Navigator (including kinematics)
- Manufacturing
- Finite Element Analysis



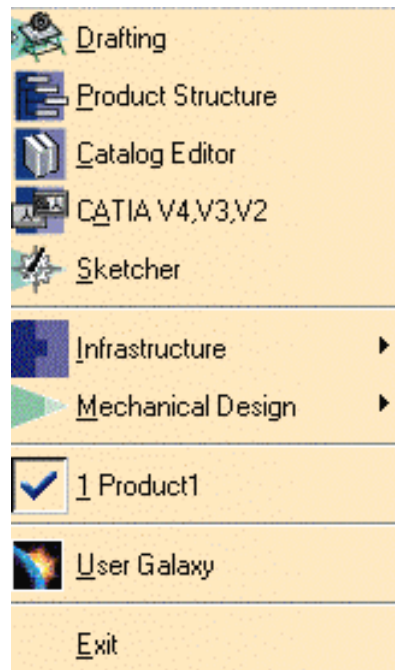
# CATIA User Interface



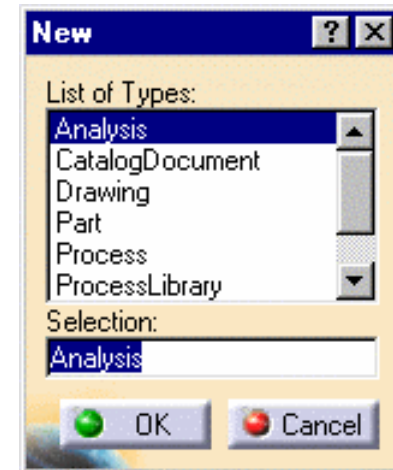
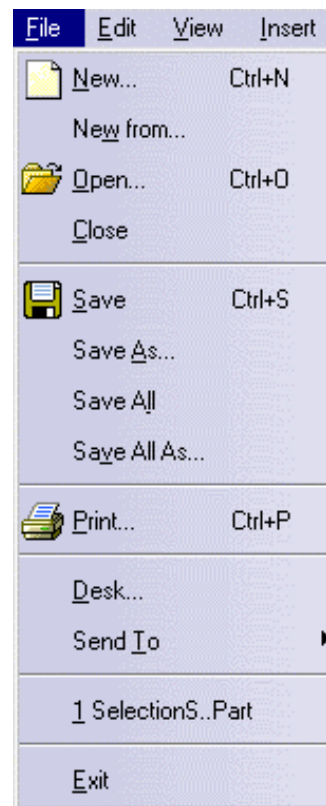


# Interacting with CATIA (1)

- Selecting an Application
  - ❑ Use the **Start** menu to select an application



- Working with Files
  - ❑ Use the File menu to create, open, save and print





# Interacting with CATIA (2)

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- Display Commands
  - Fly Through
  - Fit View
  - Layer control
  - Pan
  - Rotate
  - Zoom
  - Normal View
  - Standard Views
  - View Types: Shaded/ Hidden  
Line/ Wireframe/ User Defined
- Hide/ Show
  - Hide
  - Swap Visible Space
- Properties
  - Display Characteristics for an object are set by selecting the entity, then pressing the right mouse button and selecting **Properties** from the menu

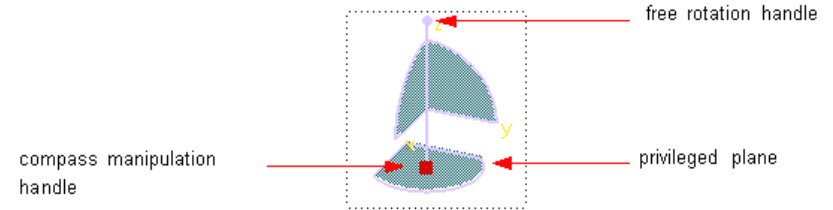




# Manipulating the Display using the Mouse

- Pan
  - ❑ Press and hold the middle mouse button and move the mouse to pan
- Rotate
  - ❑ Press and hold the middle mouse button then the left mouse button and move the mouse to rotate
- Zoom
  - ❑ Press and hold the middle mouse button and click the left mouse button then move the mouse to zoom in and out

- Using the compass



- Drag the axes or planes of the compass to dynamically rotate the display
- Multi-select entities by holding down the **Shift** key

# More Common Commands

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- Copy/ Paste



- Geometry entities can be copied and pasted from one part to another.



- Paste Special allows you to:
  - Paste a complete copy with history
  - Paste a linked copy
  - Paste the result without linking

- Undo/ Redo



- Allows you to undo previous actions



- Redo repeats an action that has been undone

- Hide/ Show



- Allows you to temporarily hide entities from the display



- Hidden entities can be recovered by clicking on the “Swap visible space” icon, and then selecting the entity to make visible

- Update

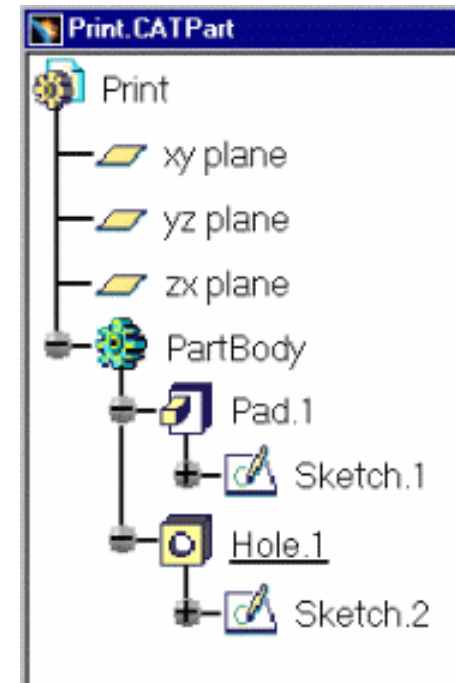


- Used to update the part after modification



# The Specification Tree

- The Specification Tree is displayed on the left side of the screen while you are working
- Provides access to the history of how a part was constructed, and shows the product structure
- Product entities can be selected from the spec. tree or in the geometry area
- Parts can be modified by selecting them from the spec. tree.
- Click on + to open a tree branch
- Solid Parts are stored in the PartBody branch of the Part tree





# Getting Help

- The online help library can be accessed by selecting the **Help -> Contents, Index and Search** command
- The Help home page provides a search facility, and allows you to browse by application.
- Every CATIA task has a getting started guide

CATIA Infrastructure Frameset - Microsoft Internet Explorer

File Edit View Favorites Tools Help

Back Forward Stop Refresh Home Search Favorites History Mail Print Edit Discuss Real.com

Links Best of the Web Customize Links Free Hotmail Microsoft Product News RealPlayer Today's Links Web Gallery Windows

Address I:\BSF\Doc\online\basug\basugfrs.htm Go

**CATIA SOLUTIONS**  
Version 5 Release 4

o pan: drag using middle mouse button. --- To rotate: pres

**Preface**

CATIA Infrastructure

Welcome to CATIA Version 5!

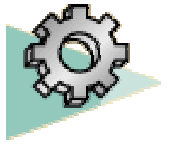
CATIA Version 5 is the first release of the next generation of CATIA Solutions, and addresses advanced mechanical process centric design requirements. In addition to leading edge feature-based design functions, it includes highly productive capabilities for the design of mechanical assemblies and for drawing generation

Preface  
What's New?  
Getting Started

# Getting Help from the CATIA Community

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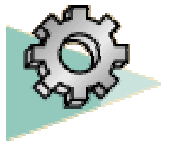
- For general information about CATIA from IBM and Dassault Systemes refer to:  
[www.catia.com](http://www.catia.com)
- For access to the database of known problems refer to:  
<http://service.boulder.ibm.com/support/catia.support/databases>
- The CATIA operator's exchange provides a forum for the exchange of ideas and advice about using CATIA at:  
[www.coe.org](http://www.coe.org)
- And look at Member Center -> Forum



# Part Design

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- The Part Design application is used to create solid models of parts
- Solid parts are usually created from 2D profiles that are extruded or revolved to form a base feature
- The Part Design task is tightly integrated with a 2D sketching tool
- A library of features is provided to allow user to add additional details to a base part
- Parts can be modified by selecting their features in the specification tree
- Parts are stored in files with the extension .CATPart



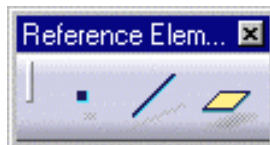
# Part Design

- Base Features



- Pad
- Pocket
- Shaft
- Slot
- Hole
- Groove

- Reference Elements



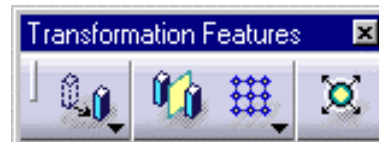
- Point
- Line
- Plane

- Dress-up Features

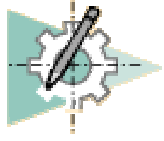


- Fillets
- Chamfers
- Draft Shell
- Thickness

- Transformation Features



- Translation
- Rotation
- Mirror
- Pattern
- Scale

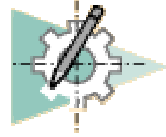


# Sketcher

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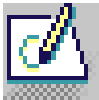
- The sketcher is used to create 2D sketches of designs, and apply constraints to the sketched geometry
- The sketcher is now the main environment for developing 2D profiles that will be used to build solid models (but traditional 2D wireframe techniques are available in the Wireframe and Surface application)
- The sketcher provides a flexible environment for creating and modifying 2D geometry





# Sketcher

- Entering the sketcher



- Click on the Sketcher icon or select Start -> Mechanical Design -> Sketcher

- Exiting from the Sketcher



- Click on the Exit icon to leave the sketcher and return to the 3D workspace

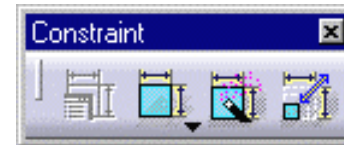
- Geometry Creation



- Geometry Operations



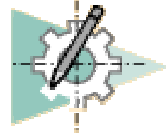
- Constraint Creation



- Tools Toolbar

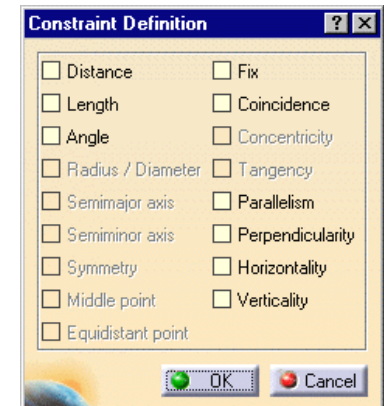
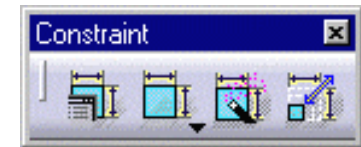


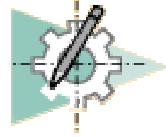
- Snap to point
- Construction Geometry
- Constraint



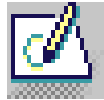
# Using the Sketcher

- The Sketcher is a parametric design tool
- It allows you to quickly draw the approximate shape of a design, and then assign constraints to complete the shape definition
- Constraints can be applied as:
  - Driving Dimensions – dimensions that control the size of a geometric entity
  - Geometric Constraints – geometric relationships such as parallel, perpendicular, tangent, collinear

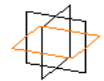
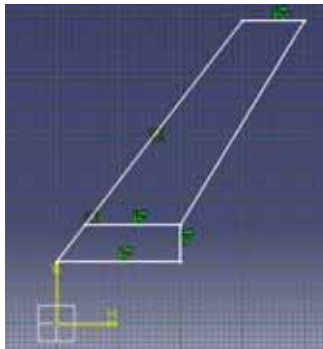




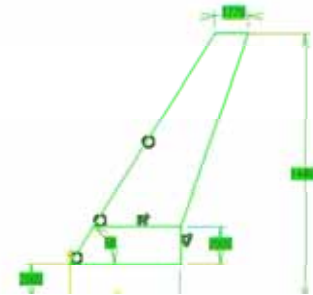
# Sketching Example



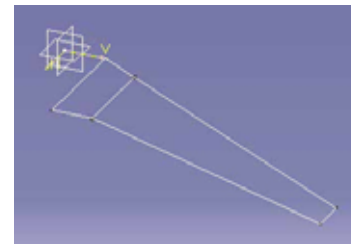
1. Click on the Sketcher icon
2. Select the 2D plane to sketch on (may be a plane, or the face of an existing part), and the sketching window will appear
3. Sketch the profile

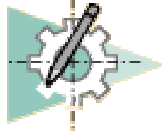


4. Apply constraints to define the exact geometry required



4. Click on the exit icon to quit the sketcher
5. Sketch is transferred into the 3D modelling environment



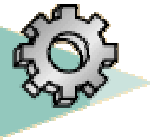


# Sketching Tips

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- To edit an existing sketch ensure that you select the sketch from the specification tree, or select an element in the sketch. (If you do not do this you will create a new sketch instead of modifying the existing one)
- If the sketch goes **purple** while you are constraining it is over-constrained. Generally it is best to **Undo** the last constraint and examine existing constraints to find the problem before continuing
- Solids can only be created from sketches that form a single closed boundary
- The profile icon allows you to create complicated profiles including lines and arcs. See the online help for more information

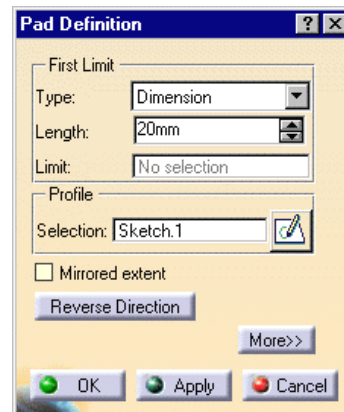




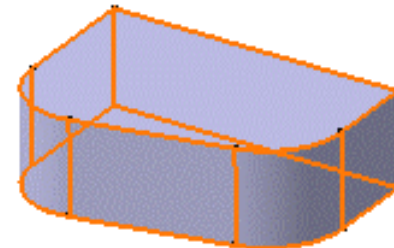
# Creating a Solid Part from a Sketch

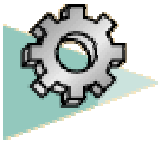


1. Click on the Pad icon to create an extruded part
2. Select the sketch containing the profile you want to extrude (note the sketch is treated as a single entity)
3. The Pad definition window will appear



4. Select the limit type from:
  - Dimension
  - Up To Next
  - Up To Last
  - Up To Plane
5. Type in the length if required
6. Check the extrude direction arrow
7. Click on **OK** to create the Part

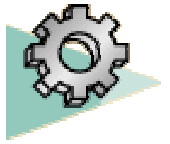




# Working with Features

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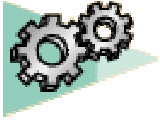
- The Part Design task uses intelligent design features
- The features contain information about their context as well as their shape
- For example a Hole feature can only be created once you have created a part body
  - ❑ A hole feature requires an attachment face, and driving dimensions
  - ❑ A hole is a negative feature – it is automatically subtracted from the main Part Body
- Other features include Pad, Revolve, Pocket, Groove, Thread, Rib, Slot, Stiffener
- When a new feature is added to a solid part it is automatically combined with the existing part



# Modifying a Part

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- All parts created in Part Design can be edited at any time in the life of the part
- The parameters used to create a feature can be accessed by double clicking on the feature definition in the product specification tree or on the part geometry
- For example to change the height of a pad you should double click on the pad node in the specification tree.
- The original feature dialogue will appear on the screen
- Change the values and click on OK.
- When you have modified the feature parameters the part will automatically update. The part turns red briefly to indicate that it is out of date

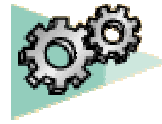


# Assembly Design

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- The Assembly Design application allows you to create a product model from a number of separate parts
- The parts in a product assembly are not joined together, but assembled as they would be in a physical assembly
- The product assembly structure is hierarchical and allows you to model complex product relationships
- Constraints can be applied between the parts in assembly to define relationships between them





# Assembly Design

- Product Structure Tools



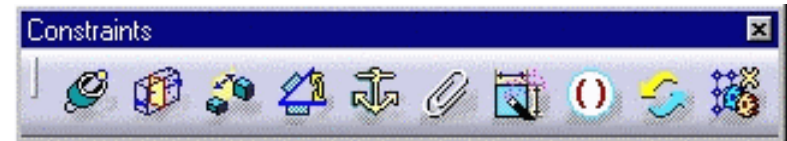
- Insert New Component
- Insert New Product
- Insert New Part
- Insert Existing Component
- Replace Component
- Reorder Tree
- Generate Numbers
- Load Components
- Unload Components
- Manage Representations
- Multi-Instantiation

- Move Toolbar



- Manipulate
- Snap
- Explode and Assembly

- Constraints Toolbar

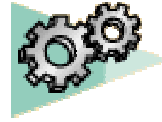


- Coincidence
- Contact
- Offset
- Angular
- Anchor
- Fix Together

# Benefits of Assembly Modelling

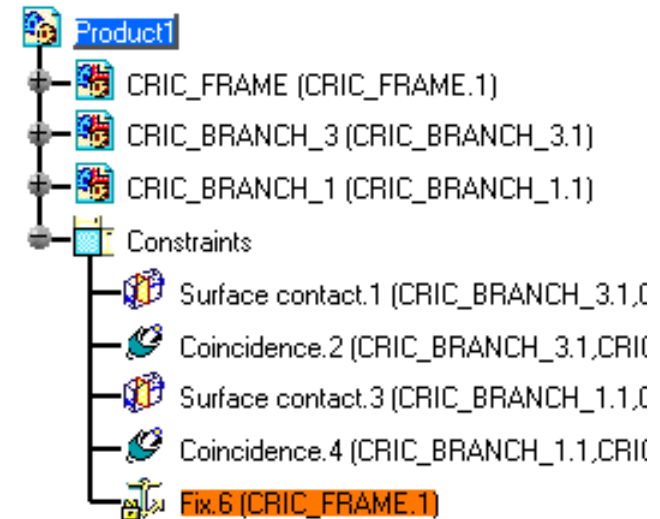
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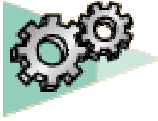
- Support for reuse of standard parts
  - ❑ Assembly design creates links to the master geometry definition, so multiple instantiations of parts can be efficiently created
  - ❑ Design changes are automatically reflected in the assembly
  - ❑ Model sizes are minimised because geometry files are not copied
- Management of inter-part relationships
  - ❑ Mating Conditions
  - ❑ Contact Constraints
- Development of Kinematics models
  - ❑ Simple mechanisms analysis available



# Using the Product Structure Tree

- The specification tree shows product structure information relating to the parts and sub-assemblies contained in an assembly
  - ❑ In the example shown on the right the product is called Product1
  - ❑ The product contains three components CRIC\_FRAME, CRIC\_BRANCH\_3 and CRIC\_BRANCH\_1.
    - The Product and the Components do not contain any geometry
  - ❑ Geometry is stored in parts inside the Component definitions
  - ❑ The Constraints Branch shows the constraints that have been created to define the relationships between the components in the product structure

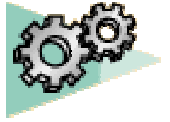




# Steps for Creating an Assembly

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1. Create a new CATProduct using ***File -> New -> Product.***
2. Use the Product Structure tools to lay out the main assembly structure
3. Use ***Insert Existing Component*** or ***Insert New Part*** to create geometry in the Assembly
4. Use Constraints to capture the design relationships between the various parts in the assembly



# Saving Assembly Information

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- Assembly information is stored in a file with the extension *.CATProduct*.
- The *CATProduct* file contains only information relating to the product assembly.
- The detailed geometric information about the parts in the assembly is referenced to the original *.CATPart* files

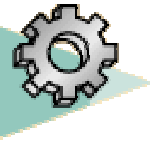
## Warning

- If you copy a *.CATProduct* file it will still point to the original part files
- To copy an entire assembly use **File -> Save All As...** , specify a new location for the *.CATProduct* file, then click on the **Propagate** button.

# More Advanced Part Design

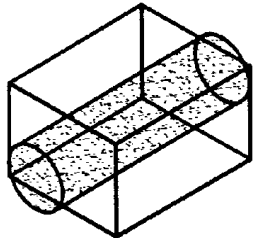
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- Boolean Operations
- Transforming Parts
- Assigning Materials
- Calculating Mass Properties



# Using Boolean Operations

## PRIMITIVES

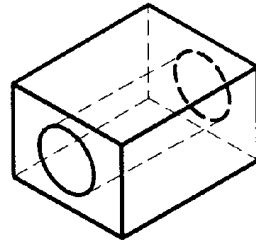


BLOCK AND  
CYLINDER

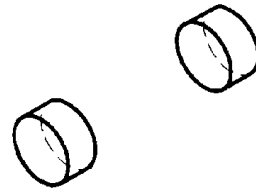
### ORIENTATION

CYLINDER AXIS  
NORMAL TO FACE  
AND CENTERED  
ON FRONT OF  
BLOCK

## DIFFERENCE

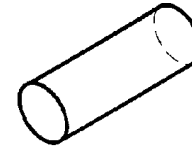


BLOCK - CYLINDER

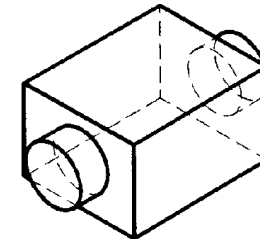


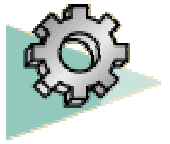
CYLINDER - BLOCK

## INTERSECT



## JOIN





# Using Boolean Operations

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- To use the traditional Boolean operations approach to solid modelling you must create multiple bodies within a part.



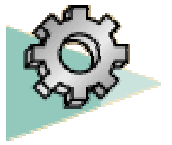
- Create additional Bodies by selecting the function

Insert -> New Body



- Boolean operations (join, subtract, intersect) can only be applied between the main PartBody, and other bodies in the same Part





# Transforming Parts

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- Solid features can be transformed using the transform functions



- Features can be mirrored, translated, rotated and scaled
- Patterns are used to create rectangular or circular arrays of features



# Assigning Materials




- To Assign a material click on the **Materials** Icon on the toolbar
- Select a material from the material library
- Click on the part you wish to assign the material to, then click on **Apply Material** and **OK**. The material will appear on the properties branch in the spec tree
- Note: You may need to change the option settings  
To make the parameters branch of the specification tree visible. To do this select

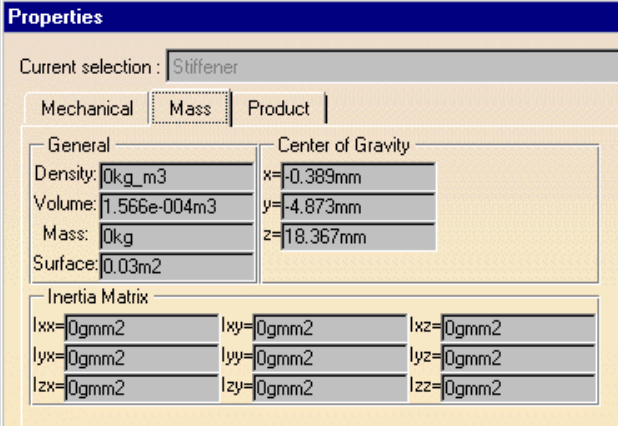
Tools->Options->Infrastructure->Product Structure  
Specification Tree -> Parameters





# Calculating Mass Properties

- Select the node of the part you want to analyse in the specification tree
- Click on the **Measure Inertia** icon 
- Or
- Select **Properties** from the popup menu on the right mouse button to see the properties form, select the **Mass** tab and view the properties:



**Properties**

Current selection : Stiffener

Mechanical | **Mass** | Product

General

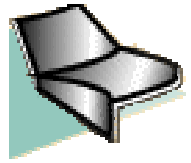
Density: 0kg\_m3  
Volume: 1.566e-004m3  
Mass: 0kg  
Surface: 0.03m2

Center of Gravity

x=-0.389mm  
y=-4.873mm  
z=18.367mm

Inertia Matrix

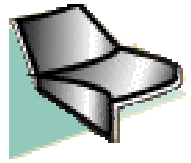
lxx=0gmm2	lxy=0gmm2	lxz=0gmm2
lyx=0gmm2	lyy=0gmm2	lyz=0gmm2
lzx=0gmm2	lzy=0gmm2	lzz=0gmm2



# Wireframe and Surface

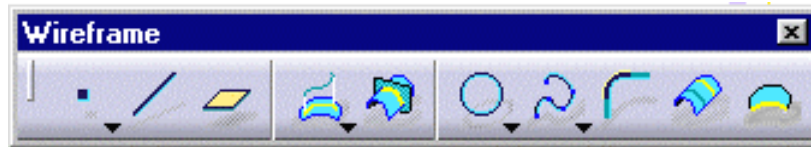
---

- The Wireframe and Surface task provides a more traditional CAD 3D modelling environment
- The Wireframe functionality allows you to create Wireframe points, lines and curves in 3D space, without using the constraint based approach of the sketcher
- The Surface functionality allows you to create smooth freeform surfaces by sweeping Wireframe curves through 3D space
- Wireframe and Surface is integrated with the other CATIA applications allowing for hybrid surface and solid modelling



# Wireframe and Surface

- Wireframe Toolbar

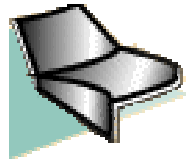


- Create Point
- Create Line
- Create Plane
- Create Projections
- Create Intersections
- Create Circle
- Create Spline
- Corner
- Create Parallel Curves
- Create Boundary Curves

- Surface Toolbar



- Extrude Surfaces
- Surface of Revolution
- Offset Surface
- Sweep Surface
- Create Filling Surface
- Loft Surface
- Blend Surface
- Extract Geometry



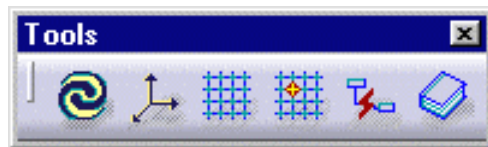
# Wireframe and Surface

- Operations Toolbar



- Join
- Split, Trim
- Transform

- Tools Toolbar

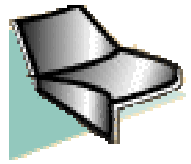


- Update
- Axis
- Work with Support
- Snap to Point
- Create Datum (deactivate History)

- Transformations Toolbar

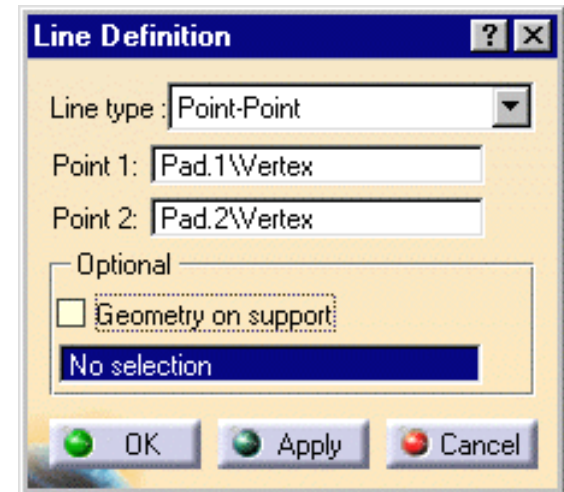


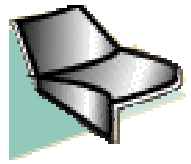
- Translate
- Rotate
- Create Symmetry
- Scale
- Affinity (irregular scaling)



# Creating Wireframe Geometry

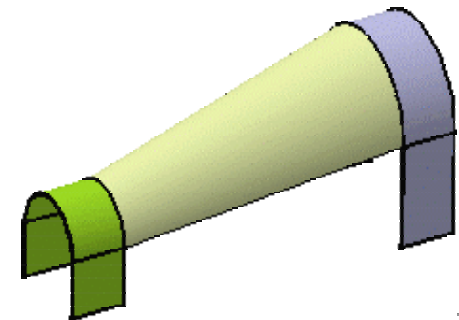
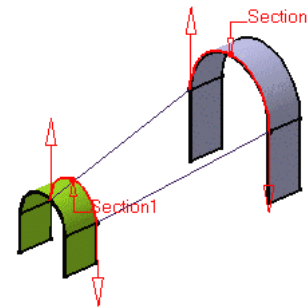
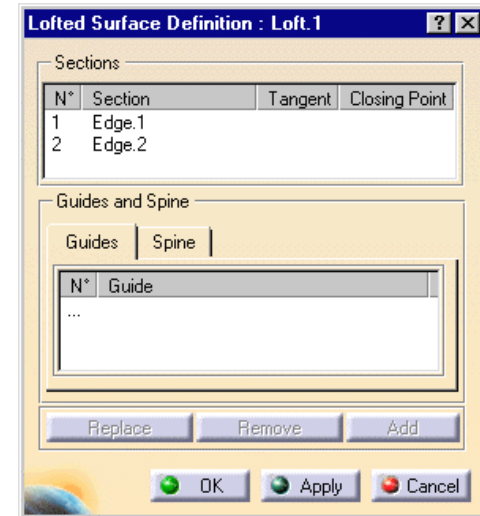
- Wireframe geometry can be created in 3D space, or on a 2D plane (using a support)
- Each wireframe function has a number of different methods (e.g. a line can be created from point to point, or parallel to an existing line, or many other ways).
- Existing geometry can be selected by picking on the screen or selecting from the spec. tree
- Additional options may be available by pressing the right mouse button over the input box





# Creating Surface Geometry

- Surfaces are usually created using a wireframe skeleton
- For example the Loft function requires 2 or more cross section curves
- It also optionally accepts a number of guide curves that extend between the cross curves
- A spine curve can be used to define the shape of the loft

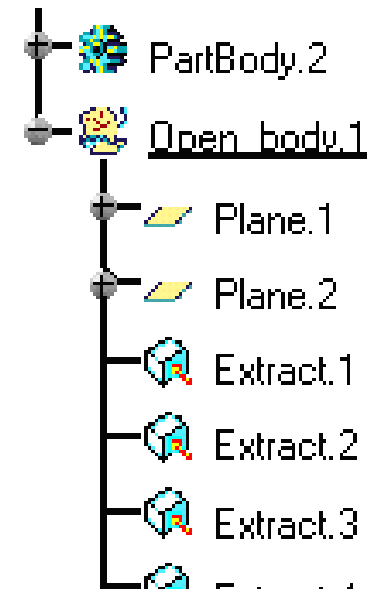




# Using the Specification Tree with Wireframe and Surface

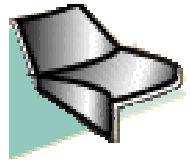


- Wireframe and Surface Geometry is created in an “Open Body” within the Part definition
- Geometry in the open body is not “attached” to the main part
- New Open bodies can be created using the Insert -> Open Body command
- A part can contain both Open Body and Part Body information

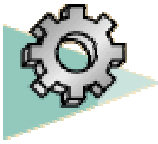


# Wireframe and Surface – Hints and Tips

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- If you want to repeatedly use the same function (e.g. to create multiple points) *double-click* on the icon. The dialogue will remain open after you click on OK.
- It can be very useful to create planes to use as a support when creating geometry.
- When creating surfaces take care that the underlying wireframe geometry is consistent, and curve endpoints are all matched
- When creating surfaces ensure that curve orientations are consistent



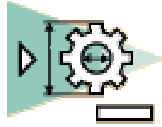
# Solid – Surface Integration

- The Part Design Application provides a Surface Based Features toolbar to allow you create solid bodies from surface models.
- Solids created from surfaces are generally more difficult to modify than solids generated in part design
- The solid part maintains associativity to the surfaces it was generated from

- Surface Based Features



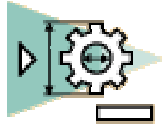
- Split – Uses a surface to split a solid object
- Thicken – Creates a solid body by “thickening” an existing surface
- Close Surface – Creates a Solid body from a closed set of surfaces
- Sew Surface – Joins a surface to a solid body



# Generative Drafting

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- The Generative Drafting Application allows you to create engineering drawings from parts or assemblies
- Generative Drafting automatically lays out orthographic projections of a part onto a drawing sheet
- Traditional Drafting functions can be used to annotate the drawing layout
- Drawings are stored in files with the extension .CATDrawing



# Generative Drafting

- Views Toolbar



- Create a Front View (other views available underneath icon)
- Create a section view
- Create a detail view
- Create a Clipping View
- Create Views Via Wizard

- Automatic Dimension Creation



- Auto-dimension
- Semi-Automatic Dimensions



# Interactive Drafting

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- Allows you to create engineering drawings without first creating a 3D part
- Provides 2D drawing functionality to create geometry layouts
- Provides dimension and dress-up facilities for drawing annotation
- Can be used to add additional information to a drawing created using Generative Drafting



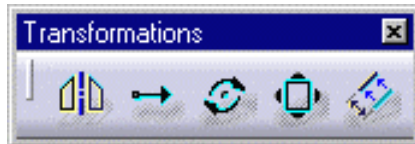
# Interactive Drafting

- Geometry Creation



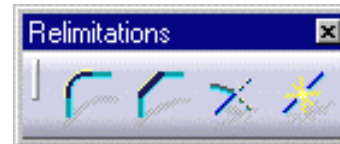
- Point
- Line
- Circle
- Arc
- Profile
- Curve
- Pre-Define Profiles

- Transformations Toolbar



- Translate, Rotate, Scale, Mirror

- Relimitations Toolbar



- Corner
- Chamfer
- Trim
- Break

- Annotation



- Text
- Symbols



# Interactive Drafting

- Dimensions Toolbar



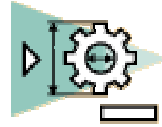
- Create Dimension
- Create Tolerance

- Dress up Toolbar



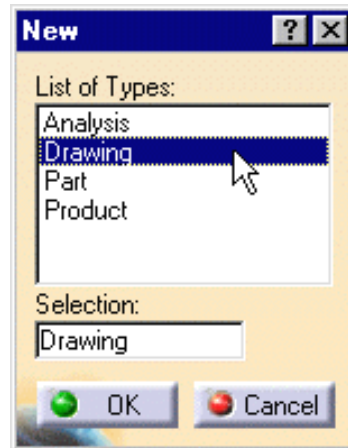
- Centreline
- Thread
- Axis
- Fill
- Arrow



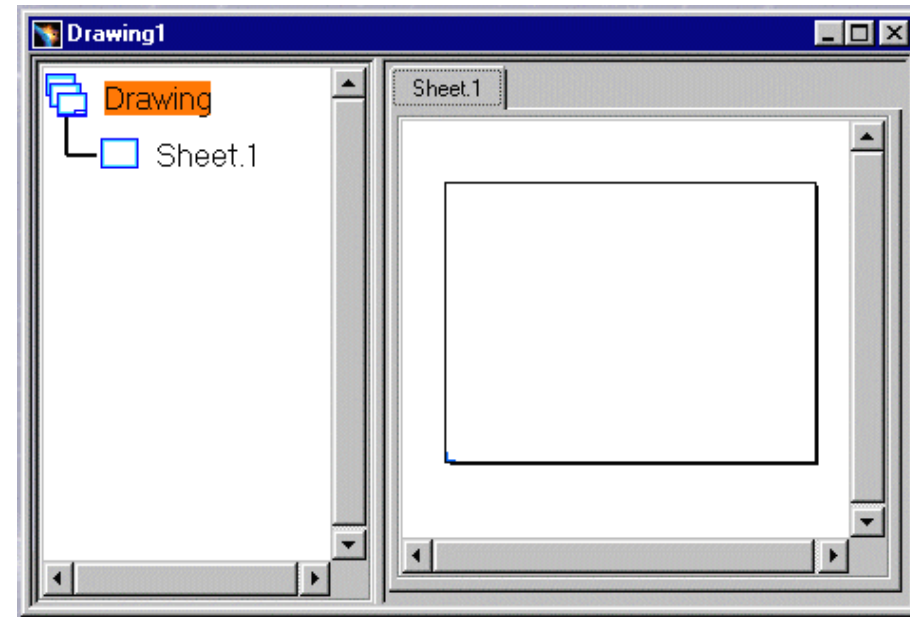


# Drafting Example

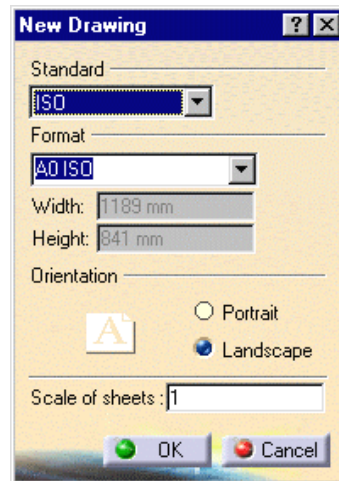
- Create a new Drawing using File -> New...

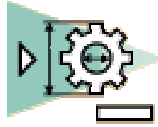


- The drawing sheet will appear on the screen



- Select the drawing Format and Scale



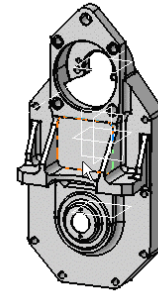


# Drafting Example

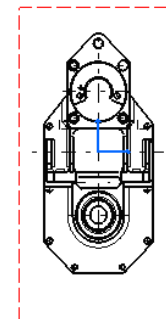
- Use File -> Open... to open the 3D part you want to generate a drawing from
- It is useful to arrange the screen so that you can see both views before continuing
- Use the View Creation toolbar to create a new view

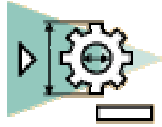


- Click on the Front View icon, then select a plane on the 3D model to specify the view orientation



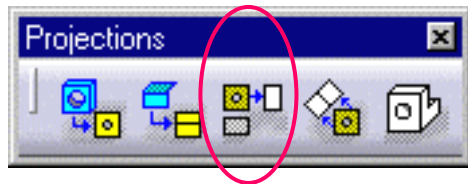
- A preview of the view will appear in the corner of the 3D window
- Click on the drawing sheet to generate the view



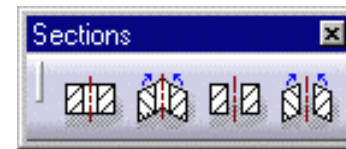


# Drafting Example

- You can generate orthographic projects from an existing view using the Projection View icon



- Sections and detail views can also be generated from existing views



# Importing Geometry from External Systems

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- CATIA provides import translators for many standard geometry formats including
  - ❑ IGES, STEP AP203, DXF/ DWG,
- Use **File -> Open** to import an external file
- The options to control the import parameters are available in
  - ❑ Tools -> Options -> Product -> External Formats (check)
- Imported CAD geometry does not contain any history information
- Check the online help for more information about the types of entities that can be translated

# Exporting CATIA geometry to other CAD systems

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- CATIA provides export translators for a number of standard formats including:
  - IGES, STEP AP203, DXF/ DWG, VRML, CGM
- Use **File -> Save As...** , then select the desired type in the **Save As Type** box to export a file in an external format
- Exported geometry does not have any history associated with it
- Check the online help for more information about the types of entities that can be translated



# Generative Part Structural Analysis

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- Generative Part Structural Analysis allows you to perform a finite element analysis on a solid part
- It is highly automated and allows an analysis to be performed with the minimum of interaction from the user
- Generative Part Structural Analysis provides very limited mesh control, and can only be applied to solid geometry
- It is generally used as a “quick check” for structural analysis



# Generative Part Structural Analysis

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- Mesh Specification Toolbar



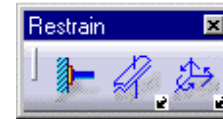
- Local Mesh Size
- Create Connections
- Create Virtual Parts

- Equipment Toolbar



- Created distributed and lumped masses

- Restraints Toolbar



- Create Clamp
- Create Slider
- Create Ball Joint

- Loads Toolbar



- Create Pressure
- Create Distributed Force
- Create Acceleration



# Generative Part Structural Analysis

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- Compute Toolbar



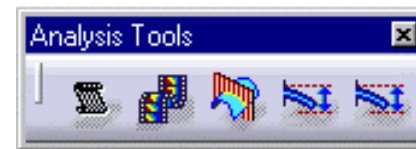
- Specify External Storage
- Compute Static Solution
- Compute Frequency Solution
- Compute Buckling Solution

- Image Toolbar



- Visualise Deformations
- Visualise Von Mises Stresses
- Visualise Displacements
- Visualise Principle Stresses

- Analysis Toolbar







# Steps for Performing an Analysis

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1. Select the parts or features for analysis
2. Define any connections, attached parts and non-structural masses
3. Specify loads and restraints acting on the part
4. Submit the job for analysis
5. Visualise Results



# Parameters and Formulas

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- CATIA V5 contains a group of applications that provide CATIA Knowledgeware capabilities
- These tools allow you to perform design automation, and capture non-geometric information about a product
- The most basic Knowledgeware tool is the Knowledge Advisor
- Using Knowledge advisor you can create parameters and relationships relating to parts



# Knowledge Advisor

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- CATIA stores information about a part in form of parameters



- Formula function – allows you to create new parameters and create relationships between existing parameters.



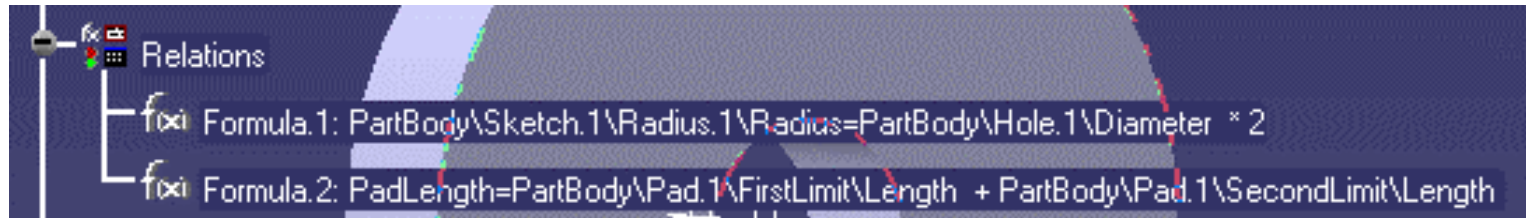
- Rules function – allows you to define design rules relating to design parameters in a part or product
- Parameters and Relations are displayed in the specification tree



# Knowledge Advisor Example

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- This relations branch shows two formulas:



- The value of the diameter Radius.1 is set equal to 2\* the diameter of Hole.1 in the part
- The value of the user defined parameter PadLength is set equal to the sum of the two limits on Pad.1