CATIA V5

INTRODUCTION:
CATIA is one of the worlds leading high-end CAD/CAM/CAE software packages CATIA programming also allows you the flexibility of using sketched and parametric based design CATIA is the leading edge technology starting with its product concept, through design, assembly, testing, manufacturing and modeling to it's rendering capabilities

EVOLUTION OF CATIA:
Developing Technical & Scientific applications by computer, first became truly viable with the advent of Second generation of mainframe in 1960's It was the aircraft industry, which first made use of the improvements driven by the critical requirements involved in optimizing airframe design For the first time, aircraft designers were able to develop programs to calculate airflow and stiffness, enabling airframe shapes to be optimized and minimized

ADVANTGES OF CATIA V5 OVER OTHER 3D MODELING SOFTWARES

- Faster design capture with solid modeling
- Faster drafting with solid modeling
- Better visualization with 3D solids
- 3D modeling reduces design errors as checking of fits and tolerances
- Associativity between Part Design, Assembly and Drafting
- Easy data transfer
- Digital mock-up unit eliminates preparation of physical prototypes
- Easy design changes are possible as CATIAV5 is a parametric and associative relationship
- Manufacturing module enables designer to understand the automated NC tool path generation and / or rapid prototyping technologies
- Most user-friendly windows based application

Formats usually read directly by CNC machines

∗stl- -stereo lithography (CATIA can transform the surface data to this format)

∗lbm- -laser beam machining

∗fdm - -fused deposition moulding

SKETCHER-1
Create a sketch
1. Start sketcher
2. Select plane
3. See workbench icon and confirm that you are entered into sketcher workbench
4. Create a profile
5. Constrain profile
6. Use mouse buttons (CTRL+MB1 for ZOOM, MB2 for PAN)
7. Use operation tools to finish sketch
8. Save the sketch
9. Exit workbench

PART DESIGNING

Start part designing
See workbench icon and confirm that you are entered into part design module

Pad
1. Create a solid (Use pad options)
2. Create a drafted filleted pad
3. Create a multi pad

Pocket
1. Create a pocket (Use pocket options)
2. Create a drafted filleted pocket
3. Create multi pocket

Dress up features - 1
Finish the part model by using dress up features
Edge fillet, chamfer, draft, shell and thick face
Use MB1+MB2 for 3D rotate

Create a shaft
1. Use different angles
2. Create a shaft by using an axis

Create a groove
1. Use different Angles
2. Create a groove by using an axis

Create a hole
Use types of hole options except threading
Understand the difference between Hole and Pocket options (Hole is a simple way to make complex features over pocket option)

Sketcher-2
(Follow same as sketcher 1)
1. Use an absolute sketcher option (Use options)
2. Use projection options (Understand the projected profiles are of yellow coloured and no constrains required because it is parametric)
3. Isolate the projected profile
4. Finish the profile by converting supporting elements to construction lines/curves
5. Use sketch analysis
6. Animate the constrains
7. Turn the view to sketch view

Create a stiffener

Create a rib

Create a slot

Miscellaneous-1
1. Use reference elements (use options)
2. Create a reference plane (use options)
3. Use an axis system
4. Turn your object to different views
5. Understand the difference between perspective and normal view

Create a lofted object
1. Understand the necessity of using loft
2. Understand the minimum inputs required for loft
3. Create a loft
4. Understand the difference between guide curves and spine curve

Create a removed loft

Dress up features-2
1. Variable radius fillet
2. Advanced draft
3. Draft with reflected lines
4. Use surface as a neutral element for draft
5. Use hole with threading options
6. Use dress up feature external /internal threading
7. Understand all threading will appear only in drafting

Miscellanious-2
1. See the show / hide zones separately
2. Understand that all sketches will automatically be hide after its 3D operation
3. Use the same sketch for other operations/constrains etc
4. Create non-parametric features
5. Understand the difference between parametric and non-parametric features

Check
Use measure options

Modify
1. Modify sketcher and all 3D operations
   Students should feel the impracticability of editing all the features
2. Use insert part body option
3. Switch between part bodies by defining them in workbench
4. Change sketch support
5. Sketch – change body
6. Change properties of each part body (use options)
7. Re-order

Boolean operations
Use Add, Remove, intersect, union trim, Remove lump and Assemble options

Transform
1. Students should aware of defining the current part body in workbench before transforming them
2. Use transformation features
3. Understand the difference between Mirror and Symmetry

Visualize
Use apply material (use types of material option)

Weekly assignment / test

ASSEMBLY DESIGN

Introduction
You can design a new product
You can make a product assembly of existing parts
You can make sub assemblies under one product

1. Start assembly designing
2. See workbench icon and confirm that you are entered into Assembly module

Design a new product
Insert parts as you require into your product
Edit each part to design
Edit product to assemble all the parts

Make product assembly of existing parts
Insert existing parts to your product
Insert sub products (sub-assemblies) to your product

Constrains
Fix at least any one part of your product which forms base of your product or which is the main part and is usually fixed
Use constrains to position all the parts
See the degrees of freedom of each part
Use manipulation for pre-positioning of parts
Use stop manipulation on clash during pre-positioning

Manipulation
Multi instantiate the similar parts
Under stand that sketch/profile projected from other parts do not have any link with its source.
Remember that projection of sketch / profile is easy and faster but modification is difficult.
Explode your assembly

1. Generate numbering items (Understand the result of numbering can be seen only in drafting)
2. Prepare Bill of materials (Use *'save as'* options)

DRAFTING

1. Start Drafting
2. See workbench icon and confirm that you are entered into Drafting module
3. Open desk to see the linking between part design, assembly and drafting modules

SURFACE MODELING

1. Start Surface modeling

2. See workbench icon and confirm that you are entered into Surface modeling
3. Refer sketcher 1&2 from Part designing module
4. Understand that closed boundary is not necessarily required for surface creation

Create primary wire frame
1. Create a plane
2. Create points, lines, polylines, circles, splines, helical curves & spiral curves

Create primary surfaces
Sphere

Create profile based primary surfaces
Extrude, Revolve, sweep, Fill and Loft

Create secondary wire frame
1. Use project, intersect, combine, parallel curves, corner & curve connect.
2. Use replication tools

Create secondary surfaces
Offset, blend

GENERATIVE SHAPE DESIGN

1. Start Generative shape deign
2. See workbench icon and confirm that you are entered into Generative shape design

Surface operations
Use surface operations to finish the surface
Extrapolate, boundary, extract, split, trim, transformation features, fillet options, join, healing, untrim surface or curve and disassemble.

Notes:
Understand the difference between point continuity, tangent continuity and curvature continuity
Also understand if the spine /guide curve is of curvature continuity then loft will be of at least tangent continuity and if spine curve is of tangent continuity then loft will be of at least point continuity
Student should feel the inconvenience of accessing a lengthy tree of operations under one open body

Miscellaneous
Insert new open body

PART DESIGN

Use surface based features in part design
Thick, split, close and sew surface
Understand that a part body can be split with the use of a reference plane also