

CATIA V5- QUESTIONS & ANSWERS

1. **Expand CATIAV5?**

Computer Aided Three Dimensional Interactive Application.

1. **What is the save extension of sketcher file?**

CAT Part

1. **Does CATIA V5 work on UNIX Platform?**

Yes

1. **Is it possible to increase the size of plane boundary representation & how?**

Yes, go for Tools-Options- Infrastructure-Part structure-Display

1. **Is It Possible to directly enter in to Sketcher Workbench?**

No, it is not possible to enter in to sketcher workbench directly. We have to go for any workbench & from there we can enter the sketcher workbench..

1. **Which is the tool used to exit from sketcher workbench to part design Workbench?**

Exit Sketcher.

1. **What _____ is use of construction elements?**

Construction elements assist in sketching the required profile in sketcher.

1. **What are the default units of LMT (Length, Mass and Time)**

mm, Kg, Second.

1. **What is SKETCH TOOLS in sketcher work bench & Explain the Importance of it?**

SKETCH TOOLS in sketcher workbench are the commands, which find very use in creating sketches. SKETCH TOOLS are namely geometric and dimensional constraints, construction elements/standard elements and Grid option. They play very important role in sketching, whenever we want to constrain a sketch we use these options and if we want to convert any element into a construction element once again these options come into picture.

1. Is it possible to hide specification tree?

Yes, with help of F3 button, but the option in Tools command must be checked to allow this.

2. What is SHOW/HIDE option?

Show mode enable us to see all the components presently opened and in Hide mode we can hide the desired elements from the view for time being.

1. What is the use of Cut Part by Sketch Plane?

This task shows how to make some edges visible. In other words, we are going to simplify the sketch plane by hiding the portion of the material that is not needed for sketching.

1. How do you measure arc length?

We can measure arc length by using MEASURE ITEM command. Sometimes we need to customize the option for arc length if it is not checked earlier using customization in MEASURE ITEM command.

1. What is the meaning of true dimension?

True dimension is the dimension desired after the machining. In other words, this is the value that should be attained after the machining.

1. What do you mean by ISO-Constraints?

If all of the degrees of freedom of geometry have been taken up by a consistent combination

of dimensions & fixed geometry,. That geometry is said to be ISO-CONSTRAINED.

Geometry that still has some degrees of freedom is said to be UNDER constrained.

16) Mention the color code of ISO-constrained, under, and over constrained elements?

The color code for these elements is Green, White and Magenta respectively.

1. What for animated constraint command is used?

This task shows how constrained sketched element reacts when we decide to vary one constraint.

1. How many dimensions are required to constrain the ellipse?

Three dimensions are required namely major axis, minor axis and the distance from the origin.

1. What are different conic sections?

Ellipse, Hyperbola and Parabola

1. What is RHO value for ellipse, Parabola and hyperbola?

Parabola has RHO values of 0.5, Ellipse has RHO value between 0 & 0.5 and Hyperbola has RHO value b/w 0.5 & 1.0.

1. What is NURBS?

Nurbs are the type of curves

1. How many types of Co-ordinate systems are there?

Three namely Cartesian, Polar and Spherical co-ordinate system.

1. What are project 3D silhouette edges?

Project 3D silhouette edges in sketcher will shows how to create silhouette edges to be used

in as geometry or reference elements.

1. What is use of sketch analysis?

To check whether the sketch is fully closed or not so that the sketch can be used or not so that the sketch can be used for further operations in part design.

1. Where do we use axis?

Axis is used in creating shaft (revolved) feature.

1. Can we redefine the sketches?

Yes.

1. Can axis be converted into line or vice versa?

We can convert line into axis but axis cannot convert into line.

1. How many axis can be created in a single sketch?

Only one axis can be created in a sketch, if more than one axis are drawn then only one of them, the latest one, will be axis and others will be converted into reference elements.

1. How do we change, sketch's reference plane?

Right click on the sketch whose reference plane is to be changed and select the change reference plane and then select new reference.

1. What is the function of mirror command in sketch?

Mirror command in sketch will create a copy of the sketch about a reference plane.

1. If I don't want the relation b/w original and mirrored elements what should I do?

Explore and the relation b/w the original and mirror element doesn't exit.

1. What is the use of isolate in sketcher workbench?

Isolated is used when 3D geometry is projected on to a sketch in order to be modified and used as part of the sketch's profile.

1. Can we select non-planar surface as sketch plane?

No, we cannot select a non-planar surface as sketch.

1. What are the different options available in quick trim command?

BREAK & RUBBER in removes part of the element, which is clicked.

BREAK & RUBBER out removes part of the element, which is not clicked.

BREAK & KEEP keeps both parts of elements after breaking.

1. What do CATIA P1, CATIA P2 AND CATIA P3 mean?

P1, P2 and P3 will indicate the different types of platforms of CATIA. Higher the number sophisticated will be the software.

1. What is kernel?

The kernel is the basic indispensable part of an operating system that allocates resources such as low-level hardware interfaces & security.

1. What is the kernel of CATIA?

CNEXT

1. Is it possible to directly enter the part design workbench, when we start the CATIA session, instead of assembly design workbench?

No, we cannot enter directly into the part design.

1. What is the importance of sketch tools?

This tool bar only appears when you are in sketcher workbench. The four tools found in this tool bar are toggle tools. When tool is highlighted the tool is on. This particular tool bar changes depending on what other sketcher workbench tool is currently selected.

1. How many degrees of freedom are there for points, lines, circles & ellipse in 2 dimensions?

Degree of freedom for points & ellipse is 2 for circles it is 3 & for ellipse it is 5 in two dimensions.

1. What is the meaning of mean dimension?

Mean dimension is the dimension that should be mean of all the dimensions, which are tolerance.

1. How many types of environment are available to start CATIA?

1. From desktop (motif)
2. From console (dterm)

2. What is hlr & nhr? What is their purpose?

Hlr = hidden line removal

Nhr = no hidden line removal

Their purpose to visualize the solids in different modes.

44) What are Master W/S and Detail W/S?

Each model can have one Master Workspace, in which the model is created & manipulated. A model can have zero or more workspaces called detail workspaces. These are auxiliary workspaces, contain elements that are to be duplicated to several locations in the Master workspace. In addition, Detail workspace can also be organized into separate Library files using the LIBRARY FUNCTION. These can be shared with models to Allow Organizational standardization

PART DESIGN

1. Expand CAD/CAM/CAE/PDM/VPM/CFD

Computer Aided (Design/Manufacturing/Engineering). Product Life cycle Management/

Product Data Management/ Virtual Product Module/ Virtual Product data management/

Computational Fluid Dynamics.

1. Is it Possible to create pocket or groove as first features?

Yes, it is possible.(body concept)

1. How to give tolerance to particular dimension?

First, give the dimension & using right click select ADD TOLERANCE from the contextual

menu & specify the tolerance.

1. What is use of creating datum?

Deactivates the link between parts.

1. Can you pad open & intersecting profile?

Possible for open profile with thin pad option. Not possible with intersecting profile.

1. Can I take portion of the one sketch for creating pad?

Yes, using the MULTIPAD option OR using simple PAD and in simple PAD select GOTO PROFILE option.

1. What is DRAFTED FILLETED POCKET?

It performs drafting, pocket& filleting simultaneously.

1. **Can we use arc as axis for creating shaft feature?**

No, we cannot use an arc as axis for creating shaft feature.

1. **What kind of profile should be there for creating stiffener?**

We can use Wire frame geometry or sub elements of a sketch. Profile may be open or closed but condition is that closed profile's extrusion must be normal to sketch.

1. **Can we give two different angles for same face of solid by using draft option?**

No, it is not possible to give 2 different angles for same face of solid by using draft option but it is possible if we use "ADVANCED DRAFT"

1. **What is power copy?**

Power copy is a set of features that are grouped under to use in different context& having the capability to adapt changes when pasted.

1. **What is user feature creation (UFC)?**

Create hybrid feature, intended to be stored in catalogues and can be instantiated later on.

1. **What is the use of the option 'Duplicate data in CATIA model' in design table?**

Check this box when you intend to reuse your document on an opening system different from the one, which is used to create the design table.

1. **Is it possible to add some more parameters to pre-existing design table?**

Yes, we can add parameter to the exiting design table with of ASSOCIATE option.

1. **What are the different options in PASTE SPECIAL?**

As result:- In this option the copied feature having neither link nor the design specification of the original one.

As result with link:- In this option the copied feature will be having link with the original one but not the design specification of the original one.

As specified in part document:- In this option, the copied feature will be having both the

link and design specification of the original one.

1. It is possible to create negative body?

Yes, using INSERT menu & INSERT BODY option

1. What is use of REMOVE LUMP?

Removing of material that is not physically connected to any body.

1. What is Reframe on & center graph?

REFRAME:- Zooms on particular object selected CENTER GRAPH Bring the selected features to the center screen in the specification tree.

1. What is the use of 'REORDER'?

The capability of REORDER command allows us to rectify design mistakes by reordering operation

1. What is the use of PULLING DIRECTION IN Rib option?

It sweeps the profile with respect to a specified direction. To select this direction, select a plane or an edge.

1. How do I create a plane at angles to another plane?

Using the option Angle/normal to plane in PLANE command.

1. What are the different types of coupling modes are there in loft?

Ratio, Verticles, Tangency discontinuity, curvature then tangency Discontinuity.

1. What is the significance of specified representation in PATTERN?

With this you can make any object invisible.

1. How I can place the instances on both sides of original feature?

Using Row 1 & Row 2 options.

1. Is it possible to pattern the two or more features at a time?

Yes (by multi selecting the features & then selecting the pattern command)

1. What is meaning of 'explode' in pattern?

Deactivating the link b/w patterns & makes them as independent entities.

1. What are all the limitations of User Features Creations (UFC) as compared with the power copy?

1. In UFC datum's cannot be used as inputs of the features.
2. Sub-elements cannot be used as inputs of the features Ex. The face of a pad cannot be used as input.
3. When creating a user features, it is not possible to edit (add/remove) inputs once you leave the DEFINITION Dialog tab. Click the CANCEL button and create the new user feature creation.

1. What is 'Keep angle' in rib & slot?

'Keep angle' option in ribs & slots lets us to keep angle value between the sketch plane used for the profile & the tangent of the center curve.

1. Which is the better option to split which a solid: - a) surface b) plane c) face?

Plane is the best option to split solid.

1. Is it possible to split using the SEW operation?

Yes.

1. What is a FUNCTIONAL SURFACE?

A FUNCTIONAL SURFACE is the element that defines the face on a solid.

1. What is IUA? What is its purpose?

IUA= Interactive User Application Its purpose is to customize the CATIA user command.

1. What is use the of MERGE END option?

'MERGE END' option when checked, will limit the extrusion to the exiting material.

1. What is the use of LAW function?

The usage of law function involves the creation of geometry to control the exiting material.

1. What are solid primitives?

Solid primitives are the ready-made features available in a particular for use. One Just needed to enter the dimensions & can have solid ready made. Example: - cylinder, cone,

sphere, etc...

1. What is 'Reference surface' option in ribs & slots?

It sweeps the profile while keeping the angle value between the axis & the reference surfaces constant.

1. Example the significance of the options 'from side' & 'from top' in creating stiffeners?

From side: - The extrusion is performed in the profile's plane & the thickness is added normal to the plane.

From top: - The extrusion is performed normal to the profile's plane & the thickness is added in the profile plane.

1. What is 'KEEP SPECIFICATION' in pattern?

By checking this option we can have instances same as that of the original & any change made in the original will be observed in the instances.

1. What is 'Simplified representation' in pattern?

By checking this we can make desired objects invisible just by clicking on them.

SURFACE DESIGN

1. What is thickness of surfaces?

Infinitely less

1. What is hybrid modeling?

Mixture of solid & surface modeling.

1. Is it possible to set default colour for surface?

Yes, we can set default colors for surface.

1. What is use of 'Federation' option in join?

The purpose of federation is to regroup several elements making up joined surface or curve.

This is especially useful when modifying linked geometry to avoid respecifying all the input

elements.

1. What is default value for distance objectives in join command?

0.001mm.

1. How does the nearest in project command will affect?

It will show its significance when there are more than one profile in a single sketch if we want to project all of them on a surface then we have to uncheck it, otherwise the only profile, which is nearer to the surface, will get projected & others will not

1. Is it possible to perform a shell operation on a sphere?

Yes, we can shell a sphere. For this, we need to just select SHELL command & give wall thickness. If we select the sphere as object to be shelled then it will show some error & we cannot shell it.

1. What is healing of geometry?

This task shows how to heal surfaces, that is how to fill any (slight) gap that may be appealing b/w 2 surfaces.

1. What are distance objectives (in healing)?

It is the maximum gap allowed b/w 2 healed elements.

1. What is 'Freeze elements' in healing?

If this option is checked, the healing operation will not affect the selected elements under 'freeze elements'.

1. What is smoothing of curves?

This task shows how to smooth a curve, i.e., fill the gaps & smooth the tangency & curvature discontinuities, in order to generate better quality geometry when using this curve

to create other elements, such as swept surfaces etc.

1. What is 'Maximum deviation' in smoothing curves command?

Maximum deviation (may be in distance or angles) is the allowed deviation between the initial curve and smoothed curve.

1. What is 'topology simplification' in smoothing curves command?

If this option is checked then it automatically deletes the vertices, thus reducing its numbers

of segments.

1. List the different commands available to create surfaces?

Extrusion, revolve, sweep, fill & multi-section.

1. What is 'simplify result' in join?

Checking this button allows the system to automatically reduce the number of elements (faces or edges) in the resulting join whenever possible.

1. What is 'Ignore erroneous elements' in join?

Checking this button lets the system to ignore the surface & edges that otherwise would not

allow the join to be created.

1. What are G0 & G1 propagate in join?

G0 propagate: - The tolerance corresponds to the merging distance value.

G1 propagate: - The tolerance value corresponds to the angular threshold value.

1. What is the file save extension of surface?

. CAT part is the file save extension of a surface.

ASSEMBLY DESIGN WORK BENCH

1. **What is PLM?**

Product Life Cycle Management: Product Life Cycle Management is the succession of strategies used by management as a product goes through its lifecycle.

1. **What is difference between Product & Component?**

Product is a collection of components. Whereas component is a collection of features.

1. **What is visualization mode?**

In this mode, only partial data is loaded to the memory of the hard disk. The data will be in the sellated form we cannot give constraints to the components in this mode.

1. **What is save extension of assembly file?**

. CAT Product

1. **What is design mode?**

In this mode the data is fully loaded to the memory & we can work on the components. The

components in visualization mode can be brought into design mode by just double clicking on the components but vice versa is not possible by just double clicking.

1. **When we use propagate directory?**

In save management, when we want to save the component files in the same file in which product is saved then we use PROPAGATE DIRECTORY. Then automatically the entire child files will be propagated to that particular directory.

1. **What is a scene? Where we use it give any one example?**

Scenes enable capturing & restoring the state of components in an assembly in a saved viewpoint.

1. **How many degrees of freedom will be there for any component in an assembly?**

Six degrees of freedom will be there for any constrained component in an assembly.

1. **In an assembly how do I measure degree of freedom of component?**

Activate the component & then go for ANALYZE Degrees of freedom.

1. What is use of stop manipulated on clash command?

It will stop the movement of component when clashed, in compass assisted movement.

1. What is the function of 'overload properties' in contextual menu?

It gives away us the option not to copy a particular component into the drafting from assembly by just selecting it (the particular component) using the contextual menu with 'overload properties'.

1. What is DESK command?

This command shows us how to view the relationship between different documents & to obtain information about their properties. (Uncheck tools>options

WHITE-loaded.

BLACK-not loaded in the current session.

RED-not been found.

1. Different types of CACHE?

LOCAL CACHE: – A read or writes directory located locally on your machine & used to store CGR files. The first time a component is inserted it is tessellated. This means that the corresponding CGR file is computed & saved in the local cache as well as displayed in the document window. The next time this components is required, the CGR file which already exists (& not the original document) is automatically loaded from the local CACHE. The user is normally responsible for the local cache.

RELEASED CACHE: - A read only cache that can be located any where on your network, several directories can be defined for RELEASED CACHE. If a CGR file cannot be found in the local CACHE, the software browses the released cache directories in their listed order to see if the CGR file is located in any of them. If the CGR file is still not found, the component is tessellated & the resulting CGR file is saved in the local CACHE. The site 'administrator' is normally responsible for the released CACHE. The default maximum CACHE size is 500MB. When the maximum size is exceeded, the automatic deletion of CGR files (on first in/first out basis) is triggered.

1. About EDIT –LINKS?

This task shows how to display the document links. Only direct links i.e.,

External documents directly pointed to by the active document can be displayed using the

EDIT-LINKS command. This activated inactive document must be activating before displaying their links. Note that you can also an element from the graph to display its links.

1. What is CSG tree?

CSG = Constructive Solid Geometry it is equivalent to specification tree in CATIA V5.

1. What CGR stands for?

CGR = Catia Graphic Representation.

ADMINISTRATION OF CATIA V5

1. How do I come to know about the release of CATIA V5?

Go to HELP About CATIAV5.

2. What is significance of CAT settings?

CAT settings play a very significant role. These are responsible for all the defaults. One can have settings according to their requirements in organization.

1. What is the purpose of IUA?

IUA= Interactive User Application, used to customize the CATIA user command.

2. What does CSG tree explain?

The CSG tree will explain the steps performed while doing a particular design, it is similar to specification tree in CATIA V5.

1. **What is MULTI MODEL LINKS (MML)?**

The Multi Model Links are functional in automatic updating of the changes made onto the part body. This can be achieved.

DIFFERENCES B/W

1. **Difference between new file and new from file?**

If you save an exiting file in another directory without changing the file name, you will only be able to open one of these files at any given time. If one of them is already open you will not be able to open the others. This is because both files have same UUID.

To avoid this happening each file must have its own UUID. This can be done by means of the File New from.

To create a new document whose basic Characteristics are same as an existing document? To do this close document you want to copy if not already closed & select file... New from... After selecting the existing document from which you want to create a new one & Click open. An exact copy of existing document is displayed with a default name. The only difference between this document and already existing one is that the new document is new UUID by File--- new from.

1. **Difference between geometrical & dimensional constraints?**

Geometric constraint is a relationship that forces a limitation between one or more Geometric elements. Dimensional constraint is a constraint, whose value determines the geometric object measurement.

1. **Difference between stacked dimension & chained dimension mode while using auto**

Constraint option?

In stacked dimension all the dimensions will be measured with respect to the reference. In chained dimension the dimensions will be measured one with respect to other.

1. **What is the difference between symmetry command and mirror command?**

In Symmetry command, the original sketch is deleted and the new one is created

About the reference plane but with the help of mirror command we can retain both the new one and the original sketch about the reference plane.

1. **Difference between trim and quick trim?**

In trim option, we can extend a line along with trimming of other unwanted elements, in quick trim we can only trim a line or curve and we cannot extend or shorten it.

1. **Difference between axis and construction elements?**

Axis is visible in part design mode and can be selected but construction elements are invisible in part design mode and are not selectable.

1. **What is the difference between spine and spline?**

SPINE: - creates a curve passing through a point on a plane & normal to one or more other planes.

SPLINE: - Creates A Curve passing through several points having tangential curvature continuity.

1. **What is the difference between ISOLATE & SEPARATE?**

ISOLATE: deletes logical link between the elements

SEPARATE: separate lines, curves & faces from their links with others.

Elements: EX:- A curve is considered as separate when it is linked to only one surface.

1. **What is the difference between PARAMETRIC SOLIDS & NON-PARAMETRIC SOLIDS?**

PARAMETRIC SOLID:- Relational model is parametric i.e. One to one relation if any change in dimension that may reflect on other dimensions.

NON-PARAMETRIC SOLIDS:- developing a solid by using surface, face, solid primitive etc, there is no one to one relation.

2. **Difference b/w PAD & MULTIPAD?**

A pad is used for single profile & multipad is used for multi profile sketch.

1. **What is difference b/w creating design table from current parameters & from pre existing file?**

1. Create design table from current parameter values: - check this option when you want to create a design table from a subset of the document parameters you just have to select among all the document parameter values.
2. Create design table from pre existing file: - check this option when you want to create a design table from the values of an external file.

1. **Difference b/w PASTE & PASTE SPECIAL?**

PASTE: - option in contextual menu enable us to simply copy and paste one location to other. But

PASTE SPECIAL: - option let the original one and us to maintained link b/w pasted feature. Any changes made to the original features, will be seen in the copied if we want & we have liberty to delink the original & copied feature.

1. **How do you differentiate positive and negative body?**

A positive body is the one which when assembled with another body it gets added and negative body is the one which when ASSEMBLED with a positive body will get subtracted & it will get added if BOOLEAN OPERATION, ADD is used instead of ASSEMBLE

1. **What is the differences b/w assemble & add /remove?**

In ASSEMBLE the nature of the bodies to be assemble are taken in to account. It means, if a negative body is assembled with a positive body it gets subtracted. But if we use add command for the same to bodies then they will get added irrespective of their nature

(+ Ve or -ve) nature REMOVE COMMAND is same as that of the ADD & thus it will not respect the nature of the bodies.

1. **What is the difference b/w affinity & scaling?**

SCALING: - resizing the body to the scale that you specify, in all the directions equally.

AFFINITY: - resizing the body the scale which you specify, in a particular directions only, specified by you.

1. What is the difference b/w join & heal?

Using join command we can join the surfaces & using the heal command we can fill the small gap b/w the surfaces.

1. What is the difference b/w save, save as, save all & save management?

Save: - using this option we save all the currently opened / modified files by old name.

Save as: - using this option using this option which are currently opened/ modified by different name other than the earlier one.

Save all: - using this option we can save all the files, even though which are not opened.

Save management: - the flexibility of the option 'save management' lies in the saving of the all files under different names & in different directories simultaneously.

1. What is the difference b/w coincidence & contact constrain?

Coincide type constraint are used to align elements, depending upon selected elements you may obtain CONCENTRICITY, COAXIALITY or COPLANRITY...to create coincidence constrain b/w a whole system their must have the same direction & same orientation in product.

Contact type constraint can be created b/w two planes, faces (directed planes)

The common area b/w two planar faces can be plane (plane contact), a line (line contact) or a point (point contact) ...

2. What is the differences b/w parametric & non-parametric modeling?

1. Relational model is parametric i.e., one to one relation. If any change in dimension that may reflects on the other dimensions.

2. Developing a solid by using surfaces, face & solid primitives etc. there is no one to one relation is called non parametric modeling.

1. **What is the difference b/w part, component & product?**

PART: - within the assembly workbench, it is either a part of the part design workbench, or; 3D entity whose geometry is contained in a model.

COMPONENT: - A reference integrated in an assembly. A component possesses characteristics related to how it is integrated in an assembly. (EX: - its relative location in an assembly).

PRODUCT: - a 3D entity which contains several components.

1. **What is the difference b/w POWER COPY & UFC?**

Parameters can be edited in power copy, which is not possible in UFC.

CATIA QUESTIONS AND ANSWERS

- **What is CATIA? What are the different modules of CATIA? What are the main Features of CATIA?**

CATIA: *Computer Aided Three Dimensional Interactive Applications.*

It is used to create three-dimensional geometric models using wire frame, surface and solid modeling constructions.

Additional application modules incorporated with CATIA provide

Capabilities for:

- Kinematics
- Robotics
- FEM mesh generation

- FEM Solutions
- NC Mill
- NC Lathe Programming
- Piping Design
- Structural Member Design and
- Image Generation

Additional Module allows data exchange between CATIA and other application and provide an internal CATIA mathematical routines and user interface.

Main Features of CATIA are:

- Maintains a full three-dimensional data base
- Allows direct construction of 3D Wire frame and Solid Module
- 3D space elements and 2D draw elements can co-exist simultaneously in the same model
- Automatic generation of machining instruction to drive an NC tool
- Geometry interface which can accept geometry from other system and analysis programs as well as extract data for delivery to other systems
- Kinematics module which simulates the movement of part in space
- Robotics module to simulate a robots work envelope

- **What are the relations b/w SURFACE, FACE, SKIN and VOLUME?**

SURFACE:

- A Surface is an infinitely thin, mathematically defined contoured area in space .It is displayed visually by isoperimetric curves and boundary curves.
- In simple terms a surface is an entity that has length and width, but essentially no thickness.
- In CATIA Surface is a Parent element for Face and Skin.

FACE:

- It is a portion on a surface defined with curves as boundaries or it is a portion in a plane defined with curves or lines as boundaries.
- Face is a child of surface

SKIN:

- A skin is a set of joined faces, surfaces, skins, or volumes, similar to the volume, but with a single domain and without closure condition.

In addition, an offset can be associated with each of the faces.

VOLUME:

- A Volume is a set of joined faces or surface or skins or volume, with total closed domains.
- While creating Volume the order of selecting the faces is important. Also a Volume can have an inner domain.

- **What is the difference between SOLID EXACT and SOLID MOCK-UP?**

SL. NO.	SOLID EXACT	SOLID MOCK-UP
1	Exact solids manipulate an exact type, that is, a non-approximated boundary representation.	Mock-up solids manipulate an approximated type B-Rep, that is, a representation resulting from an approximation of the non-planar forms by planar facets.
2	They are distinguished by having precisely defined surface definition.	They incorporate surfaces that are approximated using discrete planar facts.
3	They can be used for all other CATIA operations including NC programming.	They can be used effectively for object visualization, constructing mock-up to validate assembly operations or for kinematics, dynamic and FEM analysis
4	These models have applications in more artistic situations with highly contoured surfaces. Note: All curves are displayed as poly lines with decartelization being determined to achieve the most realistic appearance without excessive computational effort.	These models are simple but are of little value for applications requiring precise surface definition.

- **What is the difference between PRIMITIVE and FEATURE?**

PRIMITIVE:

It is generally the simplest solid elements that can be create.

The three types of primitives are:

- **Canonical Primitives:**

- Defined by *geometric values*:

Parallelepiped

Cylinder

Cone

Sphere

Torus

- Defined by *contour and geometry values*:

Revolution: elements obtained by rotating an open or

Closed profile about an axis.

Sweep: elements obtained by sliding contour along a spine while the normal to the contour plane remains parallel to the tangent to spine.

From skin surfaces or non-planar faces by

Closing through offsetting the same surfaces.

Closing through projection onto a plane.

Closing using planar faces.

- **Macro Primitives:**

Macro primitives are SOL type elements obtained from dittos (DIT type elements).

The corresponding detail has SOL type elements itself.

FEATURE:

- A feature is a set of user defined technological object consisting of:
 - Geometry
 - Parametric definition and
 - Technological attributes
- Normally, features are characterization details of a part that have a strong correspondence or linkage to a particular manufacturing process.

(Ex: a counter bored hole, a boss around a hole, a pocket)

- Features are defined by the user or the administrator, stored in the library and used to create parts of features, compound features.

- **What is the difference between SOLIDE+MODIFY+OPERATION+DUPLICATE and TRANSFORM+SYMMETRY?**

The first difference between these two operations is that with the SOLIDE+MODIFY+OPERATION, you must first indicate whether or not this is going to be duplication or replacement. The second major difference between TRANSFORM and MODIFY+OPERATION+DUPLICATE is that transform keeps a link between the originating primitives where MODIFY+OPERATION+DUPLICATE does not. In other words, with transform, if a change is made to the originating solid, the copied solids will also change.

- **What is the difference between SURF1+GEOEXTR and LIMIT2+SURFACE+EXTRAPOLATE?**

SURF1+GEOEXTR	LIMIT2+SURFACE+EXTRAPOLATE
Extrapolated surface will not be having the same deviation and degree of the original surface but is tangent only to the one side of the surface. (i.e., straight segment surface)	Extrapolated surface will be having the same Shape, deviation and degree of original surface.
Surface generated through extrapolation is separate entity with original surface and needs concatenation of surfaces.	The resultant surface after extrapolation is single surface. (i.e., automatically concatenated)

- **What is the use of part Editor Window?**

Part editor allows visualization and real time modification of your designed part by selecting or directly acting on the feature structure of the part.

- **What are contextual menu and its applications?**

Clicking on a branch or feature displays the corresponding contextual menu. This tool help you get commonly used operations faster such as:

- Color: to change the color of the solid
- Parent: to manage the parents of the element
- Delete: to delete the element (=delete no keep)
- Break: to separate one branch from the whole solid (=delete keep)
- Active/Inactive: to activate or inactivate the element

(or only fillets or drafts of the branch)

- Collapse/Expand: to reduce a whole branch to single component

(Or vice versa) such as a group of holes

- Smart/Unsmart: to active or deactivate smart solid.

Also in part editor, their two types of feature lists can be possible.

1. Simplified view of feature list:
2. Detailed view of feature list :- internal views of feature branches and macro primitives are displayed.

- **What is model?**

A Model is an individual drawing, read into main memory for interactive processing.

Model can contain one or more workspaces which in turn can contain one or 4more sets with zero or more elements in each.

- **What are the functions of FORMTOOL?**

- Form tool allows performing semiautomatic filleting, trimming and breaking operation on a shape with sharp edges.
- Form tool creates surface, face and skin in a single interaction
 - Skins are the main tool of the Form tool function since they allow us to combine faces and consider them as a single entity.
- Form tool allows creating variable radius fillet.

- **What is the use of LAW function and where is its application?**

The purposes of a law are easy to understand, but its application may be more complex. Laws involve the creation of geometry to control different aspects of a surface. A surface may be controlled by aspects of area, radius (width) or angle. Depending on the transitions that need to make another surface element, users may need to develop 2 dimension wire frame that will dictate how the transition is to be made in ratio proportion to the application of the law geometry. It sounds a lot harder than it is use.

- **What are sets?**

A set is a disjoint group of elements of different types that can be processed together

- **What is an element?**

Elements are the lowest level geometry entity created by their corresponding functions.

- **What is the session?**

Session is a set of models.

Session is used to allow two or more models to be positioned to create a more complex assembly.

A Session can contain several 'passive' Models but only a single 'Active' Model, which is the model displayed in the CATIA workspace.

A Session is defined by a set of models and a SESSION MANAGER.

The SESSION MANAGER configures a CATIA session and manages the data

Working with a session is working in context; this allows you to visualize your digital mock-up and to perform various simulations.

A session is stored in a SESSION-type file. It must be considered as a temporary work environment save.

No data transfer capability is provided on a session.

- **What is workspace? What is the difference b/w Master and Detail Workspace?**

Each Model can have one MASTER workspace which is the area in which the model is created and manipulated. In addition, a model can have zero or more workspace called DETAIL workspace. These are auxiliary workspace that contains elements that are to be duplicated to several locations in the MASTER workspace. In addition, DETAIL workspace can also be organized into separate library files using the LIBRARY Function.

These can be shared with many models to allow organizational standardization.

16. Is it necessary to break cylindrical surface along vertically for generation of faces?

No need to break the surface because the catia itself automatically create two faces along circumference.

17. What is the difference b/w SPACE mode and DRAW mode?

SPACE MODE:

a. In space Mode, it is possible but often quite awkward to work directly in the 3D Space. Ex:

Entering points that lie in a plane can be tedious when three coordinates must be entered for

each point.

b. In Space Mod e, CATIA allows the user to temporarily switch into a 2D mode to create,

view or manipulate elements.

c. The 2D Mode is very convenient for working with 3D Geometry in the Space Mode of

operation.

DRAW MODE:

1. The Draw Mode of operation is purely 2D Mode in which CATIA can be used for the drafting purpose.
2. The geometry is 2D only but can be organized into up to 255 views. Each view can

be defined by a geometry transformation with respect to another view. In this way, it

is possible to create a multi view orthographic projection engineering drawing.

3. The Draw Mode can be used independently or it can be used to project a full 3D

Model into Multiple Orthographic projection for purpose of preparing a traditional

engineering drawing. This process is referred as CATIA Draw/Space Integration and

is managed by special CATIA functions.

4. The Draw Mode is probably the most useful way to generate paper copies of a CATIA Model for engineering purposes.

18. What is the difference b/w

1. DITTO,COPY and TRANSFER options in DETAIL function
2. HELIC Pitch and Redial Pitch in SURF+REVOLUN+UNSPEC
3. PT type element and CST type element.
4. ARC and SPLINE
5. CUR1+COMBINE and CURV 1+PROJECT

CATIA V5 QUESTIONS

1. Define Explain the following?

Abbreviation for CATIA

Smart pick

Isolate

Extract curve

Manual update

Solid combine

Reordering

Surface element, Volume element, Constraint element

2. Types of: **Constraints:**

Sol: 1)
Geometrical 2) Dimensional

Limits: Coupling in Multi-section Solid:

Sol: 1) Ratio 2) Tangency 3) Tangency then Curvature

Transformation:

Sol: 1) Translation 2) rotation 3) Scaling 4) Symmetry 5) Affinity 6) Axis to Axis

Boolean operation:

Sol: 1) Assemble 2) Add 3) Remove 4) intersect 5) union trim 6) Remove lump

Draft:

Sol: 1) Draft angle 2) Reflect line 3) variable draft

Pattern:

Sol: 1) Rectangular 2) circular 3) User defined

Continuity in Extrapolate:

Sol: 1) Tangent 2) Curvature

Types of Continuity:

Sol: C0: Point continuity

C1: Tangent continuity

C2: Curvature continuity

Assembly constraints:

Sol:

Coincidence 2) Contact 3) Offset 4) Fix together 5) Angle

1)

Primitive:

Sol: 1) Canonical 2) Complex 3) Micro

3. Extension for w/b

Part	.CATPart
Sketcher	.CATPart
Surface design	.CATPart
Assembly	.CATProduct
Drafting	.CATDrawing
NC/Machining	.CATProcess
Analysis/GSA	.CATAnalysis
Catalog Editor	.CATCatalog
Material	.CATMaterial

4. Expand the following:

IGES	Initial Graphics Exchange Specification
STEP	Standard Exchange for Product Module Data
STL	Stereo Lithography
DXF	Drawing Exchange Format
CSG	Constructive Solid Geometry (Parametric)
LPFK	
NHR	
SHD Model	
NURBS	Non Uniform Rational B-spline

5. Assembly feature in Design w/b?

- Assembly Pocket
- Assembly Hole
- Assembly Split
- Assembly Remove
- Assembly Add

6. Features used in Solid Modeling

Sol: a. Geometry

b. Parametric definition

c. Technological attributes

7.

Short cut keys for:

Sol: a.
Middle-Pan

b. Middle+ Pan-Rotate

c. Middle+ Right (press & release)-Zoom in/Zoom out

8. Steps to regenerate?

Sol: Tools – Option – General - Display

3D accuracy Fixed=0.01mm

2D accuracy_fixed+0.01mm

CATIA

Absolute Coordinates: Coordinates that specify a location in relation to the current coordinate system (0, 0, 0)

Active View: A view from which you create any element another view or 2D dress up. The view from which section views, section cuts and detail views will be created. This view is generally corresponds to either the front view or the isometric view.

Affinity: An operation in which an element is transformed by applying X,Y,Z affinity ratios with respect to a reference axis system.

Aligned Section View: A section view creates from a cutting profile defined from non parallel planes. In order to include in a section certain angled elements, the cutting plane may be bent so as to pass through those features. The plane and feature are then imagined to be into the original plane.

Annotations: An entity that provides information's for the drawing Texts are annotations entities.

Approximate Mode: One of the various modes that can be used to generate views. The approximate mode is particularly well assigned to the sophisticated products or assemblies involving large amount of data. Although approximate views are not as high in precision quality as such views, this generations mode dramatically reduces memory consumption performance may also be improved.

Attribute: In the drafting workbench, the graphical and /or geometrical properties inherited from 3D element.

Back Clipping: A back clipping removes all the elements behind the pre-defined plane. It can only be applied on an extracted view. A back clipping plane is a plane used for generating a back clipping.

Background View: A sheet dedicated to frames and title block.

Basic Curve: If surface is trimmed at an arbitrary curve it is sometimes wanted that the trimmed surface yields the information above the input surface. This input surface is called Base Surface. (If it is not trimmed) A trimmed surface is called face and the underlying untrimmed surface is called Base surface. If a surface is not trimmed it makes no sense to distinguish between this surface and the basic surface. If you break it with the option geometric the result is not a face and in general the resulting surface is not meet the starting surface exact there is an approximation.

Bend: A feature joining two walls.

Bend Extremity: Axial relimitation for a straight bend.

Bezier Curve: A Bezier curve is a polynomial curve in the 3D space(X, Y, Z) Space which was transformed with a change of its basis. The new basis is the set of Bernstein polynomials.

The change of the basis creates in a canonical way a set of points. These points are called the control points of the Bezier Curve.

Bill of Material: A piece of information inserted into the active view of a CAT Drawing document. For this you can be either in the working view or in the background view.

Blend	Curve:	A
curve created to connect two pre-existing curves.		

Blend Surface: A surface created to connect two pre-existing surface.

Body: A group volumes and features combined to represent a solid part or product. Any number of bodies can be in a single model or file but only one can be active at a time. Volumes and features are automatically added to the active body.

Boundary: A Topological limit of an element.

Breakout: A breakout is a partially removed section which allows visualization of particular element in the view. A breakout view is one but in direct projection from the view containing the cutting profile. In other words it is not positioned in agreement with the standard arrangement of views. A breakout view is partial section.

Broken View: A view that allows shortening an elongated object using two guides corresponding to the part to be broken from the view extremities.

B-Spline: A B-Spline is a curve in the 3D space (x, y, z space) which contains more than one segment. Each segment can be considered as a Bezier curve. These Bezier curves are merged very well to avoid control points and knots at the segment boundaries. The parameter values at the segment boundaries are called knots. These knots can be distributed equal spaces Uniform B-spline (UBS) or arbitrary distributed Non Uniform B-Splines (NUBS)

Callout: A graphical representation of a cutting profile.

Cartesian Coordinate: The coordinates of an element defined according to the horizontal and vertical position of this element.

CGR Mode: One of the various modes that can be used to generate views. CGR (CATIA Graphical Representation) corresponds to a data formed containing a graphical representation of the geometry only which available with the visualization mode (geometry which is available with design mode). CGR views are not as high in quality as exact views but they contain much less memory during the generation. This may be useful when dealing with sophisticated products or assemblies during large amount of data.

Child view: A view generated from a parent view.

Clipping View: A view modified via a clipping profile.

Clipping Profile: A zone to be kept and visualized in a view.

Construction Element: A construction Element is an element that is internal to, and only visualized by, sketches. This element is used as positioning reference. It is not used for creating solid primitives

Control point: A control point is a point which a spline (tangent) passes through.

Cutting profile: A set of planes used to define a section view section cut.

Datum Feature: An element defining a contacting surface on a part.

Datum Target: An element defining a containing surface on a part and represented by spherical or pointed locating points.

Design

Tree:

Area of the document window reserved for the viewing the design specifications of a drawing presented in the form of a tree structure.

Detail View: A view corresponding to a zoomed particular area to be visualized is defined by a circle or a given polygon. This view is computed using a Boolean operator from the 3D.

Drawing: The root feature. Sheets are aggregated in the drawing. Views are aggregated in the sheets.

Dress up: A graphical attribute of a 2D element.

Design Table: It provides you with a means to create and manage component families. These components can be for example mechanical parts just differing in their parameters values.

It is a tool intended to ease the definition of mechanical parts. It is provided to all CATIA users. But you will make the best use of it in a Knowledge Advisor application. A design table can be created from a CATIA document the document data is then exported to the design table. It can also be applied to a document the document data is then imported from the design table.

It is designed to drive the parameters of a document from external values. These values are stored in the form of a table either in a Microsoft or excel file on windows or in a tabulated text file. When using a design table the associate the right document parameters with the right table parameters. The design table columns may not all document parameters and you may decide to apply only part of the design table values to you document associations. You declare what document parameters you want to link with what table columns.

It becomes a more powerful tool when it is used with knowledge advisor. You are provided with functions to create design table parameters. These design table functions can be used when programming your checks and rules. Using these functions spares you all the associations operations.

Exact View: One of the various modes that can be used to generate views. Exact views is generated from the design mode i.e. they are views for which the geometry is available.

Explode: An operation that gives 2D objects depth (3D)

FD and T View: A view that is extracted from a 3D part that is assigned 3D tolerance specification and annotations.

Feature of size: Geometric shape defined by a linear or angular dimension which is a size (ISO 14660)

Filter: A restriction on elements to be cut in a section view or section cut or elements to be seen in a projection view.

First Angle Projection method: An orthographic representation of the views comprising the arrangement around the principal view of an object of some all the other five views of that object. With reference to the principal view the other views are arranged as follows the view from above is placed underneath the view from the below is placed above. The view from the left is placed on the right and then the view from the rear is placed on the left or on the right as convenient.

Flange: A feature is created by sweeping a profile along a spine. The different flanges or swept walls available are simple and swept flange hem and tear drop.

Fleed component: A component for which all degrees of freedom are locked in relation to the parent component.

Front view: A projection view obtained by drawing perpendiculars from all points on the edges of the part to the plane of the projection. The plane of projection upon which the front view is projected is called the frontal plane.

Front plane: A plane of projection upon which the front view is projected.

Functional modeling: Refers to designing a 3D digital model by using tools with inherent behaviors such as features and volumes that interact in specific ways.

G0: If the end point of curve k_1 meets the end point of curve k_2 then we say: at this point both curves are connected with order of continuity G_0 .

If one edge of the surface s_1 meets an edge of the surface s_2 then we say along this edge both surfaces are connected with the order of continuity G_0

If the G_0 continuity is missed then we have a so-called G_0 error. This error is an absolute error, a distance and it is measured in mm or inches.

G1: The curve k_1 and curve k_2 are connected with the order of continuity G_0 in the point P . If both curves in the point P run into the same direction, this means the angle between the tangents of both curves is 0, and then we say the order of continuity is G_1

The surface S_1 and surface S_2 are connected with the order of continuity G_0 along the curve C we take the normal of S_1 in a point near the curve C and run with this normal over the border to S_2 . If the normal does not change its angle from one point of the border of S_1 to the nearest point of S_2 then we say the order of continuity is G_1 .

If the G_1 continuity is missed then we have a so-called G_1 error. This error is an absolute error an angle and it is measured in degree of rad.

G2: The curve K_1 and the curve K_2 are connected with the order of continuity G_1 in the point P . we look at the curvature vector of K_1 in point P and the curvature vector of K_2 in point P . If both vectors have the same direction and the same absolute value, then we say the order of continuity is G_2 .

The surface S_1 and the surface S_2 are connected with the order of continuity G_1 along the curve K . If each curve on S_1 , which runs over the border to S_2 , can be continued with another curve on S_2 and the order of continuity is G_2 then we say both surfaces are connected with the order of continuity G_2 .

If the G2 continuity is missed then we have a so-called G2 error. This error is a relative error and it is calculated with the following formula K1 may have the radius R and K2 may have the radius at the common point, with $r < R$, then yields

$$\text{Error} = 2 \cdot (R - r) / (R + r)$$

The maximum of this error is 2. Sometimes this error is measured in percent then its maximum is 200%.

G3: The curve K1 and the curve K2 are connected with the order of continuity G2 in the point P. for the definition of the G3 continuity we look at the curvature hedgehog, as it can be created with the command porcupine curvature analysis. We look at the envelop of the curvature hedgehog. If this envelop has at the desired point G1-continuity then we say the order of continuity between both curves is G3.

If the G3 –continuity between both curves is missed, the G1-continuity of the envelope is missed then we have a so-called G3-error between both curves. This error is an absolute error, an angle, and it is measured in deg of rad and it is the G1 error of the envelope G3 continuity between surfaces is defined on the curves between both surfaces on the same way.

Gaussian Curvature: The gaussian curvature is calculated from the Max. principal and the minimum principal curvature with the following formula.

$$\text{Gauss} = \text{sig}(\text{maxprinc曲率}) \cdot \text{sig}(\text{minprinc曲率}) \cdot \text{sigabs}(\text{maxprinc曲率} \cdot \text{minprinc曲率})$$

Sig is the sign (of maxprinc曲率 and minprinc曲率) and can only have the value +1 or -1

Generative view style:-A set of pre defined parameters and options which let you customize the appearance behaviour of the generative view.

Global deformation: - A deformation that is applied globally to a set of elements, as opposed to a deformation successively applied to a different elements.

Grid: - There are commands, which have in their properties panel the option Translate Grid.

If Grid is ON and the Grid value is not 0, then it is impossible to snap to points, which are not on the Grid.

Example:- If the Grid value is 25 then it is only possible to snap to points with the distance of 25 mm in each coordinate.

We have an Absolute Grid, short Grid. The Absolute Grid has a Grid point at the origin of the model Coordinate System. It can be switched on with Translation, Grid.

The other Grid is the Relative Grid. The Relative Grid has a Grid point at its starting point of modification. The Relative Grid can be switched on with Translate Discrete.

Healing: - The action of filling a gap that may exist between two adjacent surfaces.

Iso-Curve: - An Iso-Curve is a curve on a surface. One parameter, u or v, runs from 0 to 1 and the other parameter is constant. Iso is the prefix for constant. For example isobar.

Iso-photo: - an Iso-photo is curve on a surface. All points of this curve of the iso-photo have the same illumination from a given light source. The illumination of all points of this curve is constant. Iso is the prefix for the constant. For example Isobar.

The topological operation in which adjacent surfaces can be assembled to make up one element.

Last Component: - The last component at the end of each branch of the specification tree.

Locked View: - A locked view is a view in which any graphical modification of the generated 2D elements is forbidden.

Loft Surface: - A surface that is obtained by sweeping one or more planar section curves along a spine, which may be automatically computed, or user defined. The surface can be made to follow one or more guide curves.

Model: - A CATIA Version 4 model.

Match curve: - A curve deformed so as to connect another curve, while taking the continuity type into account.

Match Surface: - A surface deformed so as to connect another surface, while taking the continuity type into account.

Mesh Line: - A line on surface used to deform this surface according to various laws, and types of deformation.

NUPBS: - A NUBS, Non Uniform B-spline is also called NUPBS; to make it more clear that it is a polynomial curve not a rational curves.

NURBS: - A NURBS, Non Uniform Rational B-spline, is a NUBS with a rational component. Rational means that the weight of the control points must not have the value 1. With a rational curve a Circle and A Hyperbola can be described exact.

Object: - In the drafting workbench there are two kinds of object Activated and Selected. The view frame of an activated view display red.

Offset Section view/Cut: - A section view created from a cutting profile defined with several parallel planes. In sectioning through angular objects. It is desirable to show several features that do not lie in a straight line by offsetting or bending the cutting plane.

Overlay: - In a multi-model context all passive elements are called over layed elements.

Overrun: - A part of a dimension is corresponding to the extended extension line.

Parent: - A status defining the genealogical relationship between a feature or element and another feature or element for instant the pad is parent of a draft.

Part: - A 3D entity obtained by combining different features.

Part Body: - A component of a part made of a combination of several features.

Pattern: - A set of similar features repeated in the same feature or part.

Pocket: - A feature corresponding to an opening through a feature. The shape of the opening corresponds to the extrusion of a profile.

Polar coordinate: -The coordinates of an element defined according to the radius and the angle of this element.

Product: - A 3D entity, which contains several components.

Profile: - An open or closed shape including arcs and lines created by the profile command in the sketcher workbench.

Power copy: - It creates set of features (geometric elements, formulas, constraints, and so on) that are grouped in order to be used in a different context. You can completely redefine these entities when you paste them. As it capture the design intent and know how of the designer, it enables greater reusability and efficiency. We recommend you to use this command for bodies, features, and sketchers and design tables that require new specifications.

To benefit from the best level of performance in the long term, use this capability to enrich your feature catalogs.

Unset breakout: - An unspec breakout operation removes locally a 3D part. It allows visualizing the inside of a 3D part. It can only be applied to an extracted view.

View Frame: - A square or rectangular frame that contains the geometry and dimensions of the view.

Volume: - The solid material in a catpart document. It can also be the inside of a shelled solid volume.

Wall: - A feature created by adding thickness to a profile.

Wireframe element: - Elements such as points, lines or curves that can be used to represent the outline of a 3D object.

The parts building the symmetrical sub-assembly are:

- Either a symmetrical part from the source part. This involves creating a new part, outside any assembly context, with a Part Number. A typical example is the left door in a car, relatively to the right door.
- Either a new instance of the source part. In a position symmetric to the original part. A typical example is a car's front left wheel relatively to the front right wheel
- FORM >> associatively: A change in geometrical shape of the source part leads to update the symmetrical part.
- POSITION >> associatively: A change of relative position of a component of the source sub-assembly leads to update position of the symmetrical component in the symmetrical sub-assembly.
- STRUCTURE >> associativity: A change in structure of the source sub-assembly (Adding/removing components) lead from the structure of the symmetrical sub-assembly.

It is necessary to restructure components by moving components from one assembly to another assembly. Sub-Assembly is a Sub-assembly whose child components can be moved disregarding the fact it is not the component. Relative positions of its child components can be different than those stored in the reference CAT product.

There are two types of structure when you use flexible sub-assemblies.

Product structure tree shows which assemblies and sub-assemblies parts and constraints belong of mechanical structure tree show what components you can constrain together (they are at the same level). Flexible sub-assembly does not exist anymore in mechanical structure tree.

Components and constraints of flexible sub-assemblies are considered as direct Childs of the root assembly in mechanical structure tree.

Once the sub-assembly is flexible, numerical value, and activity status. Orientation (same or opposite), Driven/driving pro be overload to modify locally its internal definition, or deal with under/over-constrained situations.

When u apply an over loaded position result: all rigid instances should have the same position than the flexible one position of flexible instances are not impacted by the command.

Desk commands shows you how to view the relationships between different documents and to obtain information about properties.

The colors used to identify the various document types are the following ones

- White for loaded documents
- Black (reverse video) for documents that are not loaded in the current session
- Red for documents that have not been found.

When the design table is created, the rank of the columns fits the rank of the parameters in the inserted parameters that you want to have columns ordered in a given way in the design table, you must insert the parameters one by one.

Accessing the functions related to the design table

Once in the formula (rule of check) editor, select the design table item in the dictionary, the list of the methods that can be applied to a design table is displayed. Select a method, and then click F1 to display the associated documentation.

In slots or ribs the depth of the profile must be equal to or less than the radius of the center curve.

A coupling tab in the loft and remove loft functions to compute the loft using the total length of the sections (ratio) or between vertices of the sections or between the curvature discontinuity points of the sections or between the tangency discontinuity points of the sections.

Tangency mode: uncoupled tangency discontinuity points are represented by a square.

Tangency the curvature mode: uncoupled tangency discontinuity points are represented by a square. Uncoupled curvature

Discontinuity points are represented by an empty circle

Vertices mode: uncoupled vertices are represented by a full circle

Sew surface: used to glue a surface feature to an existing 3D solid.

3D constraints can be used whenever you have 3D geometry that you wish to link to some type of 3D datum plane or surface. They are also useful when you need to drive the location of a piece of geometry created earlier in the design from a geometry created later in the model. Thus this will limit some of the need to re-ordering of the part.

Note: this capability will allow you to drive location of features in the tree from features created after them without having to the location of features in the tree.

It is possible to create a local axis in order to define local coordinates. For example, it is, sometimes, easier to build a point by coordinates in a local axis rather than creating it in the absolute coordinates system.

A flag note with leader can be attached to a part in order to give information for example on surface treatment. This flag is at hyperlink that can start any documents such as a presentation, a Microsoft excel spreadsheet or a html page on the

When creating dimensional constraints, you can define a tolerance. Using the mean dimensions icon you can compute the mean dimensions and the part will be updated. This can be useful for a part to be machined scanning a part means to replay the construction history of a part and isolate temporarily any feature to work locally. The parts of the relationships provide an accurate view of genealogical links between elements. Parent children command lets you the features isolate is used with 3D geometry is projected into a sketch in order to be modified and used as part of the sketch's profile, isolate duplicates the element since the original element cannot be changed since other geometry depend on break used to divide an isolated element into two parts at a specified point (usually to use one side of this element in the sketch). Assembling/adding: If body2 is assembled or added to body1, the operation between the bodies is a union. The only difference between the two is that assemble will respect the nature of features. If body2 contains as its first node a pocket feature (permissible), assemble will see it as a pocket and remove material from body1. In this case, if add is used, the pocket will be seen by body1 as a pad.

Intersecting: the resulting material is the intersection between the two bodies

Union Trim: The Union Trim is basically a Union with an option to remove or keep one side or the other. In the picture on the right, the purple face is selected to remove the right side and the blue face is selected to keep only the topside. For the unions trim to work, the geometry must have sides that are clearly defined.

Remove Lump: All the above options work between two bodies. The remove lump works on geometry within a specific body. If a single body has material that is completely disconnected, each piece of disconnected material is defined as a "Lump". The user can delete any lump as a single entity even if the lump is a combination of numerous features.

After certain operations, it may happen that some lumps or cavities appear in the part. We need to remove them. The remaining lump command allows you to remove lumps and cavities.

You can copy a sketch in a document then paste it into another document keeping the link with the first one. You can use the copied sketch and in case of modification of the original sketch the document in which the copy is used will be also modified.

A component is the general terminology. It can be a part or an assembly (inside an assembly it is called a sub-assembly).

An Assembly or product is a collection of components and constraints them. Assembly documents have the file extension.

CAT Product.

Parts and assemblies have a Part Number (the name of the component).

All instances of a part or assembly have the same part number. Each instance can have its own instance name that the instance.

The active item is the item currently being edited. You make it active by double-clicking on it.

Blank sheet behind the component icon means that the component is linked with an external file.

Fix is like fix in space, but when constraints are updated, it will only stay at its current place and will not go back to a "fixed space" position.

Provides four conventional standards for tolerance:

- ASME: American Society for Mechanical Engineers
- ANSI: American National Standards Institute
- ISO: International Organization for Standardization
- JIS: Japanese Industrial Standard.

Also provides three CATIA-CADAM standards:

- CCDANSI: CATIA-CADAM American National Standards Institute
- CCDISO: CATIA-CADAM International Organization for standardization
- CCDJIS: CATIA-CADAM Japanese industrial standard

Publishing geometrical elements is the process of making geometrical features available to different users. This operation is useful when working in assembly design context.

A power copy is a set of features (geometric elements, formulas, constraints and so forth) that are grouped in order to be a different context, and presenting the ability to be completely redefined when pasted.

This power copy is a template that works at the part level. From a collection of features (geometry, literals, formulas, constructions and the user can create his/her own feature. The result is a part design feature or a shape design feature that can be reused for the design of another part. The created feature can be saved in a catalog.

Features:

- Allows to create applicative features
- Allows to hide design specifications and preserve confidentiality (for instance to sub-contractors)

Create Datum :

Shows how to create geometry with the history mode deactivated.

In this case, when you create an element, there

The stiffener definition dialog box is displayed

Two creation modes are available:

From side: the extrusion is performed in the profile's plane and the thickness is added normal to the plane.

From top: the extrusion is performed normal to the profile's plane and the thickness is added in the profile's plane you cannot select the view containing the table.

The view must be in the same drawing

If you modify the 3D part and update the drawing, the view in the table will be updated as well.

Formulas are features used to define or constrain a parameter. A formula is a relation: the left part of the relation is the parameter to be considered; the right part is a statement. Once it has been created, a formula can be manipulated like any other feature from its contextual menu. The formula language uses operators and functions of all types whereby you can carry out operation parameters.

A formula is a feature, which is assigned a parameter called the activity. The activity value is a Boolean. If the activity is set true, the parameter value cannot be calculated from the formula. If a formula is created for a parameter, which is not already constrained by another formula, the activity of the new formula is set to true by default.

A parameter can be constrained by several formulas, but only one formula can be active at a time. Before activating a on a given parameter, you must deactivate the other formulas defined on the same parameter.

The incremental option of the formula editor

The incremental option allows you to restrict the list of parameters displayed in the dictionary.

Only the first level of objects right below the selected feature will be displayed in the dictionary if the incremental option is unchecked, all the objects below the selected feature are displayed.

Incremental mode is useful when you work with large documents and when the parameter lists are long.

About the formula editor: you write a formula, the formula editor provides you with a dictionary. This dictionary exposes the list of parameters and you can use to define formula. Depending on the category of objects to be referred to in the formula, the dictionary is two or three parts. To insert any definition in the formula editor, just double click the object either in the dictionary or in the double click a function in the dictionary, its signature is carried forward to the formula editor.

It provides you with a means to create and manage component families. These components can be for example mechanical parts just differing in their parameter values.

Is a tool mainly intended to ease the definition of mechanical parts? It is provided to all CATIA users. But you will the best use of it in a knowledge advisor application. A design table can be created from a CATIA document; the document data is then exported to the design table. It can also be applied to a document; the document data is then imported from design table.

Is designed to drive the parameters of a CATIA document from external values. These values are stored in the form of table either in a Microsoft excel file on windows or in a tabulated text file. When using a design table the trick is to associate the right document parameters with the right table parameters. The design table columns may not all correct to your document parameters and you may decide to apply only part of the design table values to your document. By creating associations, you declare what document parameters you want to link with what table columns.

Becomes a more powerful tool within it is used with the knowledge advisor. You are provided with functions to design table parameters. These design table functions can be used when programming your checks and rules. Using functions spares you all the association operations. To know more, click here

Here is a good example of mechanical parts that can be described by a design table. To simplify, imagine they are all checked by four parameters: the head width, the head height, the body width and the body height. The sets of four parameters that can be assigned to a screw can be easily regrouped in a design table. This design table has as many columns as screw parameters and as many rows as sets of parameter values in a design table, a set of parameter values is called a configuration and it is registered in a row.

A design table can only be created from non-constrained parameters i.e. from parameters, which are neither referred in an active design table nor used, in any other active relation.

If you keep the activity option checked for design table0 and you try to create another design table. You will have to set the parameters to add to your second design table among a restricted parameter list. Uncheck the activity option if you try to deactivate a design table and reuse its parameters in another design table.

Any time you modify a design table, the relations that refer to this design table detect the modification and turn to updated status.

As long as a design table is active, the parameters, which are declared in it, are constrained parameters and you allowed modifying them.

Double-clicking a design table in the specification tree displays the design table with its set of configurations and allows selecting a new configuration.

Only parameters, which are not already constrained by any other relation or by any design table, can be used to create a design table. If a parameter is already constrained, it does not appear in the parameters to insert list in the table dialog box.

Selecting the parameters to be inserted in a design table.

The filter name and filter type filters can be used to restrict the display of a parameter list. If you specify X in the filter name field of the select parameters to insert dialog box. You will display all the parameters with the letter x in their

ADDITIONAL QUESTIONS

1. What is spline, did you see it in your studies?
2. What is polynomial degree of curves in V4 and V5?
3. What are all the earlier name given to CATIA?
4. What is the basis for CAD software classification?
5. What is the use of form tool function?
6. What are the negative modeling concepts?
7. What are TRANSLATORS, explain?
8. What is class-A surface***
9. What is the use of ADJUST command?
10. What are different mode licenses available?
11. What is CATIA session?
12. Whether the Curves have orientation or not?
13. Whether segment lines can be made into un limited lines?
14. How many types of transformation can be achieved in CATIA?
15. Describes any two types of surface modifications?
16. What is the difference b/w the Bezier spline and Bezier Curve? Which one will be
17. The best to use and why?
18. What is major difference b/w SURF2 surface, Net Surface and NURBS Surface?
19. Where and when these surfaces have applications?
20. Explain the design procedure of the Free Form Design with example?
21. How do you check the accuracy of the surface?
22. How to access the Parent Element of the model using Contextual Menu of the part Editor Window?

23. What are the different modeling methods on CATIA? Which one you choose the best method?

Infosys Questions:

- 1.
2. What is harness?
- 3.
4. Which one you would feel compatible b/w solid model and surface model?
- 5.
6. If we give you various section contours of different size with 10mm intervals. How you are going to generate a surface model using this data?
- 7.
8. What is the difference b/w surf 1 and surf2?
- 9.
10. How do you generate surface model of stiffener?
11. Methods/ways of creating:
12. Chamfer
13. Corner
14. Trim
15. Break
16. Fillet
- 17.
18. Difference between the following
19. Project & Intersect 3D elements
20. Reverse side & reverse direction
21. Keep angle in Rib & Slot
22. Thick & Thin solid
23. Edge & Face blend
24. Tangency & Minimal
25. Positioned & Non-positioned sketch

26. Draft angle & isoclines taper
27. Taper & draft
28. UDF Pattern & Group
29. Group & group feature
30. Reference & Render sets
31. Setback blend & callout
32. Thick surface & thickness
33. Reframe ON & Centre Graph
34. Remove face & Replace face
35. Part body & Open body
36. Healing & Joining
37. Power copy & UDF
38. Symmetry & Mirror
39. Snap & Smart move
40. Crv & Ccv (curve & composite curve)
41. Broken view & break out view
42. Spline & Spine
43. Surface, face & Skin
44. Models & Files
45. Volume & Element
46. Text & TextD2
47. AUX VIEW & AUX VIEW2
48. Warm & Quick start
49. Connect & Corner in Surface w/b
50. Detail & Quick detail view
51. Analysis, Relative/Absolute
52. ANSI, ISO, & ISO representation
53. Section view & Section cut
54. Limit2-Face Break/Divide
55. Curve1-Project/Combine
56. Model/Session
57. Limit1-Concatenatie-crv/ccv
58. Limit2-surf Extrapolate/Surf-co-Extrapolate
59. Point Projection/Limit On/Off
60. Clip & BREAKOUT in Auxiliary -view2
61. Neutral & Parting Element in Draft
62. Cliff & Rolling Edge
63. Erase Workspace/Current
64. Data & Details
- 65.
- 66.
67. Reordering in Part design?
68. Contextual menu of 'Font'-Pitch
69. How to export / Import dwg to other units?
70. How to create /remove TEMPLATE/FRAME?
71. How to align isometric dwg dimn?
72. Why half-dimn gives double value?
73. Can u create a Shaft using a line?

74. Creating a sphere?
75. What is Hold-curve in Face-to-face fillet?
76. Drafting w / parting element
77. Creating text in part modeling & maintaining its orientation?
78. Switch OFF, TRIM Manually
79. Methods of solving geometry?
80. How to find intersection point of 2 curves?
81. Degrees of freedom, ISO-constrain?
82. How many Axes can be created for a sketch?
83. How to activate Alphanumeric window in CATIA?
84. How do you access CSG?
85. Working MODES in CATIA?
86. Methods of starting CATIA?
87. Hierarchy of starting a complex solid design?
88. Give the multi-section option for a Face, Volume & Skin?
89. What is a Master workspace? How do you create additional workspaces?
90. What is functional surface?
91. What is Dynamic sketcher? Where is it used?
92. Use of compass in DS?
93. Sketcher colors convey a meaning more than the same colors Name & explain?
94. What is planar face? How does it differ from face created from surface?
95. How does surf2 function differ from surf1 function?
96. How is surface element represented in v5?
97. Purpose of feature based design
98. Explain briefly about the commands used in FBD?
99. What is local function window? Explain.
100. How is the window divided in assembly?
101. What are the contextual commands available on
102. Constraints
103. Parts
104. What are the parameters that can be defined in a law?
105. What are the different curves required to define a radius law?
106. What are the 4 principal standard available in CATIA for dimension?
107. Use of LOCK in drafting?
108. How is updating done in a solid?
109. Define a constraint?
110. What is unclosed profile?
111. Can unclosed profile be extruded?
112. What does ANCHORING the profile do in sketcher w/b?
113. In how many ways you can select XY Plane?
114. The actual process of extruding a profile adds what branch F3?
115. List two different methods to delete an entity?
116. What three things must be selected to create a hole that is accurately located in a part?
- 117.
118. How would U modify:
119. The diameter of a hole that is already created?
120. The location of a hole that is already created?

121. V5 gives U several methods to rotate a part. (Changing the parts relationship to 0,0,0 point).

List 2 of them.

1. How can specification be expanded to show additional branches?
2. What is default color of the part while be updated?
3. What 2 things must be selected in order to use the rotate command from the Transformation

features?

1. What are the mode options available while creating chamfer?
2. What are the steps to create a New body?
3. What are the similarities b/w Part design & Drafting workbenches?
4. What determines the size of fillet when TTF is used?
5. Describe the process to subtract one body from another?
6. What must be selected before components are inserted into the Assembly w/b?
7. What tool is used to move the parts in Assembly w/b?
8. How do you split the screen so that u can see the cat Drawing & Part design screen at a

time?

1. List 2 different methods of moving the parts in Assembly?
1.Manipulate & Compass
2. What are the 5 options available to create a line?
3. What tool must be selected to see the hidden objects?
4. What are the 6 options for creating a plane?
5. What tool is used to a solid (Part body) from a surface (Open body)?
6. In GSD which tool is used to define the part thickness?
7. What is profile in relationship to Sweep tool?
8. What is Guide curve in relationship to Sweep tool?
9. A plane is required to complete the sweep operation?
10. How could the DMU navigator be used in Product Review Process?
11. What tool inserts a part or assembly file into the DMU navigator w/b?
12. How do you move the horizontal ground grid if it is activate?
13. Difference b/w Fly Mode & Walk Mode?
14. Basic equations about Surf2, Patch and Blend surf?
15. What is tangent continuity/ curve continuity? What are the methods to check them?
16. Is it possible to sew a SURFACE to a SOLID? -----YES
17. In a dummy solid there is a hole of 50mm. Is it possible to fill that hole?
1.Soln: Yes, Using Thickness or Remove face
18. In a sketch if 2 closed profiles are intersecting with each other. Is it possible to extrude either of the curves? -----YES
19. In assembly w/b are there any options other than constraints to position the parts, or products?----- 1.Snap

2.Smart move

3.Manipulate

4.Compass

20. What is the operating system on which CATIA works?-----UNIX & WINDOWS
21. Can you change a BODY to a Part body?
22. Is it possible to re-order the tree in assembly?-----Graph Tree Re-Ordering
23. How many parts can we select at once in Graph Tree Re-ordering?-----ONE
24. How to change the units?
25. How to change/perform:
26. Change sheet size & Angle of projection?
27. Align dimensions, Re-route?
28. Represent hidden line, Centerline, Axis line?
29. Add/Remove leader, Break point, Interruption?
30. Fake dimensions, hide/show frame
31. Section line direction, Arrow type?
32. Pickable in drafting?
33. Half dimension, Stacked dimension, Cumulative dimension, folding lines?
34. Deactivating annotation, tables & rows
35. Front view using
 - 1.Local axis system.
 - 2.Selection sets
36. View generation mode?
37. Indicate Horz/Vertical dimm for circular object?
38. Copy & Paste?
39. Inserting BOM, Background view, CGM?
40. Insert sketch dimensions?
41. Insert Dia. symbol, Sub & and super script, Hole dimension?
42. Define the following:
43. Driving dimension?
44. Selection callout?
45. Advanced front view?
46. Types of:
47. Views
48. Selection view
49. Constrains
50. Any possibility of increasing
51. Can individual sketch be saved?
52. How do you change the drawing units?
53. List all the geometric symbols?
54. What is sewing?
55. How to remove clipping in the view?
56. What is healing geometry & freeze elements in Healing?
57. What is the use of Remove Lump?
58. What is Reframe on and Center Graph?
59. What is the use of PULLING DIRECTION in Rib Option?
60. How to create a plane at an angle to another plane?
61. What is the different type of coupling modes in loft?
62. What is simplified representation in pattern?
63. How I can place the place instances on both sides of original feature?

64. How do pattern the two features at a time?
65. What is meaning of explode in pattern?
66. Explode option does what in mirror feature?
67. What is the meaning of rows in direction in pattern?
68. What does thick option do in pad command?
69. Is it possible to create a pad along any direction?
70. Difference between pad and multipad?
71. What is design table?
72. What is association and disassociation design table?
73. What is the use of Duplicate data in catia model in DT?
74. What is Resolve in CATLOG?
75. What is Thickness? Can we use contoured face for thickness?
76. What is split surface? can we use contoured surface for splitting
77. What is Sew surface?
78. Difference between sew surface and split surface?
79. What is close surface?
80. How do we give different draft angles from neutral element for a face?
81. What is the difference between square and cone in draft command?
82. Is threads visible in part design?
83. Is it possible to change the size of plane rep?
84. What is Replace?
85. What is Remove surface?
86. What is combine solid?
87. What is create datum?
88. What is mean dimension?
89. What is current operated solid?
90. What does keep edge option does in edge fillet?
91. What is user Pattern?
92. What is reframe on and define work in object?
93. What is save extension of sheet metal part?
94. What is selection sets?
95. How do you create pentagon or any other regular polygon?
96. What is Output feature? How it will be helpful?
97. What is publication?
98. How axis can be published?
99. What are the different types of parameters available in catia?
100. What is use of cut part by sketch plane in sketcher
101. What is Auto search?
102. Expand CAD/CAM/CAE/PLM/PDM/VPM/VPDM/CFD
103. Is it possible to create pocket or groove as a first feature?
104. How to give tolerance to particular dimension?
105. What is the meaning of menu dimension?
106. What is the use of create datum?
107. Can you pad open and intersecting profile?
108. Difference between PAD and MULTIPAD
109. Can you take portion of the one sketch for creating PAD?
110. What is DRAFT FILLETED POCKET?
111. Can we use ARC as axis for creating shaft feature?
112. What is difference between from top and from side option in stiffener?
113. What kind of profile should be there for creating stiffener?

- 114. Can I give two different angles for same face of solid by using draft option?
- 115. What does merge end's in rib option do?
- 116. What is user feature creation?
- 117. What is difference between create design table from current parameters and from existing

file?

- 1. What is POWER COPY?
- 2. Is possible to add some parameters to existing design table?
- 3. Difference between POWER COPY & UFC(User Feature Creation)
- 4. Difference between PASTE & PASTE SPECIAL
- 5. Different option in PASTE SPECIAL
- 6. Is it possible to create negative body?
- 7. How do you differentiate positive and negative body?
- 8. Difference between assemble and add, remove.