Catia v5 part modelling tutorial pdf

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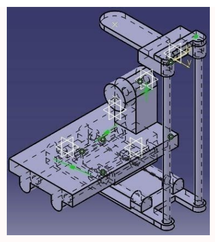

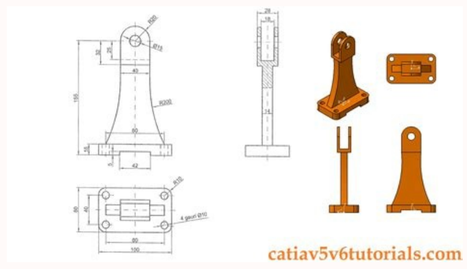

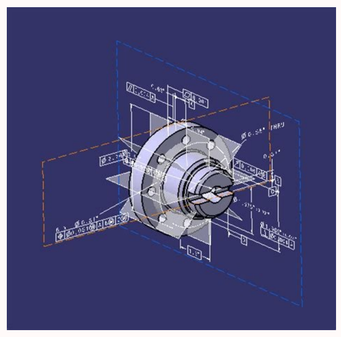

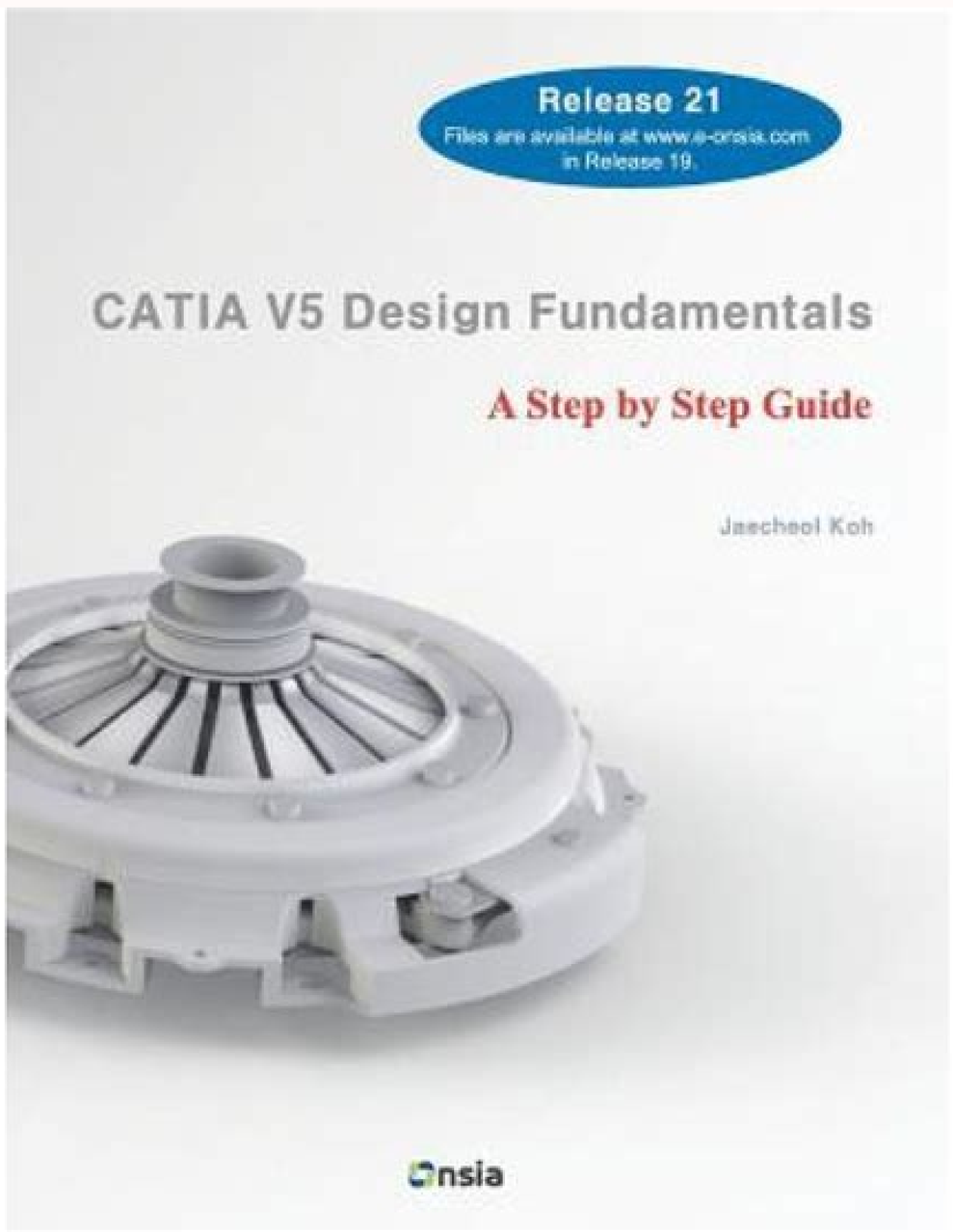

CATIA V5-6R2015 Basics introduces you to the CATIA V5 user interface, basic tools and modeling techniques. It gives users a strong foundation of CATIA V5 and covers the creation of parts, assemblies, drawings, sheetmetal parts, and complex shapes. This textbook helps you to know the use of various tools and commands of CATIA V5 as well as learn the design techniques. Every topic of this textbook starts with a brief explanation followed by a step by step procedure. In addition to that, there are tutorials, exercises, and self-test questionnaires at the end of each chapter. These ensure that the user gains practical knowledge of each chapter before moving on to more advanced chapters.Table of Contents1. Getting Started with CATIA V5-6R20152. Sketcher Workbench3. Basic Sketch Based Features4. Holes and Dress-Up Features5. Patterned Geometry6. Rib Features7. Multi Section Solids8. Additional Features and Multibody Parts9. Modifying Parts10. Assemblies11. Drawings12. Sheet Metal Design13. Surface DesignIf you are an educator, you can request an evaluation copy by sending us an email to online.books999@gmail.com Figure 1-The Angle Bracket part’s modelling stage Create features that depend on external inputs first This may seem obvious but most models are not started this way. The external inputs are driving factors for the part’s modelled geometry, therefore should be addressed first. This will help to use them as the driving elements of the parts design. Sketches are always picked from the Skeleton The sketches have to be defined in a geometrical set, outside the PartBody. This makes it easier to check the model’s specification tree and correct the sketches when necessary. Apart from holes, that create a sketch automatically to define the centre point, all other sketch-based features will let the user pick an existing sketch. The reasons why we should have the sketches in the Skeleton geometrical set have been discussed in previous articles. One profile per solid feature Apart from multipad and multi pocket tool that have multiple domains by definition, the features defining solid geometry must use a single domain of a sketch (if you have several domains in a sketch and you still want to use them then you should use the sub-elements of a sketch option). Using a single domain will guarantee that you will add or subtract one volume of material to your solid with each feature. This way you will never have two parallel extrusions controlled with a single feature; this would be confusing to analyse and harder to edit at a later stage. Define features using reference elements Many features can use reference elements to define height or depth instead of a dimension. Use reference elements instead of dimensional control because they help implement design intent and minimize the number of required modifications when editing a part. Apply additive features before subtractive features An additive feature can overlap other additive features without issue when working on the same body and the same is true when we overlap subtractive features. We need to be careful not to refill a cavity, or the open side of a cavity when we apply additive features so we always apply subtractive features after we apply the additive ones. Use the most adequate axis system to define your geometry A part can have as many axis systems defined in it as required. This can be useful for referencing to other parts at assembly level and useful to define geometrical elements using a more suitable, local, axis system. Fix all features with warnings or errors By the end of our modelling stage, we will have features that may have warnings or even errors and cannot be resolved properly. An unresolved element is an entity that cannot be updated properly therefor; all of these must be addressed and sorted out before a part can be considered complete. Delete deactivated features Deactivated features represent geometry that is not being used to define the model. For that reason, they must be deleted. If they are not deleted, these features can be reactivated at a later stage and create update issues in the model. Master parts are the exception to this rule; they can have deactivated features because they are never sent to production and can be developed to create part families; some components may have different specification tree within the part family. Do not recolour geometry in red or orange Red is CATIA’s diagnostic colour, it applies it over elements that need updating and orange is the colour it applies over selected elements. Applying any of these two colours to your geometry may cause confusion or mislead other users that may need to use models you made. Avoid hollow voids in parts It is perfectly possible for CATIA to create and model hollow parts. We need to be careful with these because hollow parts are complicated to manufacture and will require special manufacturing processes which are much more expensive. When editing a part with subtractive features and if these are not positioned or defined properly, it is very easy to have situations where the designer has a subtractive feature inside the solid geometry, thus creating an accidental hollow void. These voids must be eliminated and the subtractive feature needs to be defined properly. Never use undo features An undo feature is a feature that is applied to remove the geometry defined in the model by other features. In many situations with complex models, it is faster and easier to apply an undo feature on a model than it is to backtrack and correct the original features to get the desired final geometry. Applying undo features creates heavier models, because we are adding additional features to the tree and creates a snowball effect when it comes to editing because we make the model even harder to edit in a future situation. However, undo features are essential to work with imported bodies and cannot be avoided in these situations because an imported body has no features, so cannot be edited. Decompose features Many geometrical features allow multiple input selection and affect all of them. When you define them this will work just fine, however when you need to edit the part at a later stage, complex features are harder to edit and are more brittle. In both cases, because of the number of inputs they are manipulating, this happens often with fillet features. For this reason, it is recommended to decompose a complex feature into multiple, simpler, features. In this article, we discussed rules for feature definition in modelled parts with CATIA V5. In the next article, we are going to discuss some additional rules to take into consideration in part files. This tutorial is for 3D printing enthusiasts who are already familiar with CATIA and would like to know the tricks for making a 3D printable model with CATIA.Throughout this tutorial, you will learn the best practices for modeling, correcting and exporting an object to be 3D printed and we will share our special tips and tricks with you. By the end of this tutorial, you’ll have mastered:CATIA software is a modeling program geared toward the modeling of industrial objects, but it can be used for much more than just that. Its function relies on the use of waves and NURBS for the base of its functions. Unlike many other programs, CATIA does not rely on a single function of meshing, which relies entirely on flat surfaces giving the impression of curves, instead it relies on a system of nurbs which take the average of those edges for a truly smooth surface. The system allows for a higher level of precision than other methods. Instead of polygonal modeling, CATIA relies on dimensional sketches which make measuring and resizing the object much more accurate. With a system based on these parameters, exporting becomes quick, easy, and surprisingly light as each individual file understands a large number of nurbs.Though this type of system can make the creation of specific and organic forms (like a face) somewhat complicated. It also limits importable files to only other wave-based files (.iges and .step more specifically).It is possible to export directly from CATIA to .STL with a high level of control and precision. This gives users the opportunity to model for 3D printing directly from the program, provided a couple of rules are respected.You do not need to be an expert or know all of the minute details of CATIA to understand this tutorial. However, it is important to be comfortable with the basic elements of the program and its user interface as this tutorial focuses specifically on the best practices for creating a 3D printable model, and not on getting started with this CAD software. Thus this CATIA V5 tutorial does not go into the basic principles of the program and is not an introduction to CATIA.For more information on beginner CATIA tutorials, feel free to check out the CATIA channel onYoutube to find a video tutorial for beginners or a complete online documentation that will help you learn CATIA V5.CATIA is paid professional software. It is possible to obtain a license from Dassault Systèmes, CATIA’s creator. There is also a student edition, available whatever the school/university you come from, for a fee of 99 euros with your student card, quite useful for learning CATIA for students. This page uses frames, but your browser doesn't support them.

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