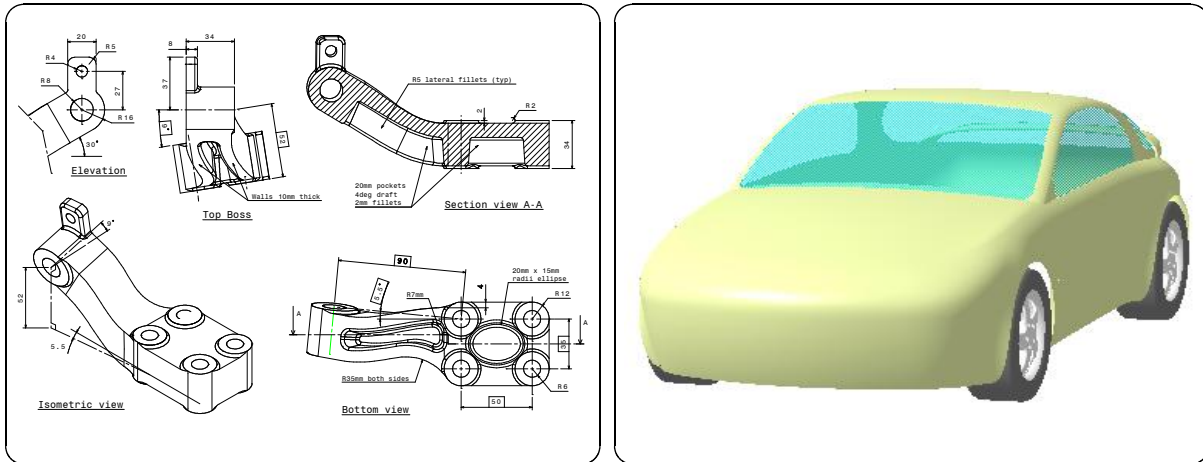


The CAD/CAM Partnership

The leading independent CATIA specialist in the UK

CATIA V5 Advanced User Guide



- ★ The CATIA V5 Advanced User Guide is the second in the series of User Guides providing genuine independent insight and easy to follow step-by-step illustrated worked examples, and also forms the basis for a **5-day** course.
- ★ Describes more advanced Sketch, Part Design, and Drafting concepts, and introduces new topics, such as Assembly Analysis, Interference and Sectioning, Bills of Materials, Surfaces, Layers, Formulas and Design Tables, and 'PowerCopies'.

The main examples include the generation and modification of a Pressure Chest Casing assembly, a complex Engine Bearing Arm which includes twisting and drafted pockets, and the surfacing and modification of a simplified concept car body, as illustrated above.

- ★ The CAD/CAM Partnership CATIA V5 Advanced User Guide answers, for example...

What happens when either a Positioned Sketch or a Sketch Support Plane is modified ?

How can Sketch Output and Profile features reduce the number of Sketches required ?

How can Drawing Number texts be created (for example, both in the title block and also top left of the sheet) so that they remain identical, and what are Catalog Text Templates, and how can they be used to annotate drawings ?

What are the effects on dependent geometry when modifying an Axis System ?

How can I create an exploded view of an assembly which will automatically reflect changes to the constituent Components ?

How can I place an Excel spreadsheet on a drawing to redefine multiple dimensions and other values, according to a specified member of a family of parts ?

What are 'DLNames' and what is the purpose of file Search Orders ?

How do I dynamically review a designated cross-sectional area of a Part, while reducing the pocket wall thickness to meet a specified weight limit ?

How are Geometric and Ordered Geometric Sets used within a Hybrid Design Part ?