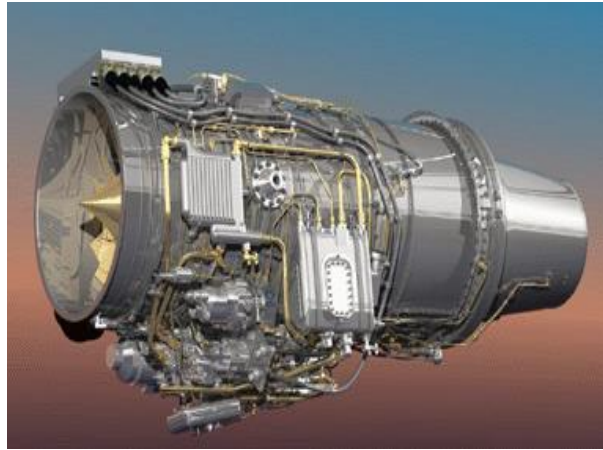




CATIA V5-R21

➤ Part Design

- Ⓞ Introduction
- Ⓞ Creating Parts
- Ⓞ Getting parts geometric information
- Ⓞ Sketcher mode
- Ⓞ Dimensioning
- Ⓞ Driven dimension
- Ⓞ Sketch planes
- Ⓞ Pad, Shaft, Rib features
- Ⓞ Parametric Modeling
- Ⓞ Modifying parts using History
- Ⓞ Adding Fillet, Chamfer, Shell and Draft
- Ⓞ Creating Patterns



➤ Assembly Design

- Ⓞ Creating Assembly Model
- Ⓞ Defining Assembly constraints
- Ⓞ Interface, Std parts, Materials etc.
- Ⓞ Bill of material



➤ Generative Drafting

- Ⓞ Generative Drafting
- Ⓞ Plotting various views
- Ⓞ Exploded Views and BOM
- Ⓞ Dimensioning
- Ⓞ Adding sectional, Detail, Auxiliary views
- Ⓞ Annotations, Symbols and Datum features, FCS

➤ Wireframe & Surface Design

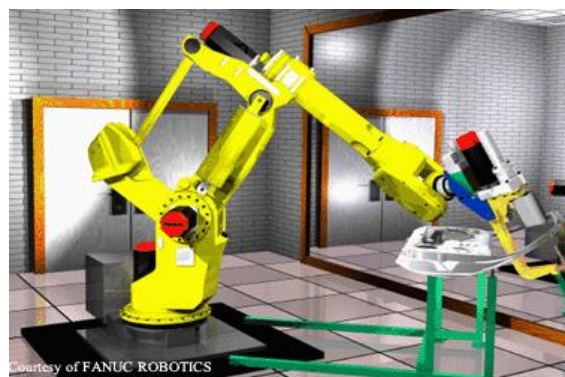
- Ⓞ Creating multisection features
- Ⓞ Creating advance sweep, fill, boundary features
- Ⓞ Modifying features
- Ⓞ Advanced surface modeling

➤ Sheet Metal Design

- Ⓞ Introduction to Sheet Metal
- Ⓞ Bend, Jog, Dimple and Flange features
- Ⓞ Hole cut, Lofted Flange features
- Ⓞ Creating Flat Pattern

➤ Extra Features

- Ⓞ Import, Export using Solid-Edge
- Ⓞ Solid-Works File Management
- Ⓞ **Blue Print Reading.**
- Ⓞ **How to apply Tolerances using IS Standards.**



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DURATION : 2 Months

