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CATIA V5 6R2015 BASICS PART II PART MODELING EBOOKS 2019



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This book is a continuation of CATIA V5-6R2015-Part I. The topics covered in this book are listed below:

- 3D Elements
- The Plane command
- Offset from plane
- Parallel through Point
- Through three points
- Through two lines
- Through point and line
- Through planar curve
- Normal to curve
- Tangent to surface
- Equation
- Mean through points
- Coordinates
- On curve
- On Plane
- On Surface
- Circle/Sphere/Ellipse center
- Tangent on curve
- Between Line
- Additional options of the Pad and Pocket commands
- Limits
- ThickView commands
- Measure Commands
- Holes and Dress-Up Features
- Hole Simple Hole
- Counterbored Hole
- Countersunk Hole
- Tapered Hole
- Threaded Hole
- The Thread/Tap command
- The Edge Fillet command
- Limiting element(s)
- Blend corner(s)
- Variable Radius Fillet
- Chordal Fillet
- Face-Face Fillet
- Tritangent Fillet
- The Chamfer command
- Draft Angle
- Draft Reflect
- Line Variable Angle
- DraftShell
- Patterned Geometry
- The Mirror command
- Rectangular Pattern
- Circular Pattern
- User Pattern
- Scaling
- Affinity
- Rib Features
- The Rib command
- The Slot command
- Multi Section Solids
- The Multi-sections Solid command
- Types of the Cross-sections
- Couplings
- Spines
- Guides
- Relimitation
- Removed Multi-sections Solid
- Additional Features and Multibody Parts
- Stiffener
- Solid Combine
- Multi-body Parts
- Creating Multi-bodies
- Insert in new body
- Assemble
- Add
- Remove
- Intersect
- Union
- Trim
- Remove Lump
- Modifying Parts
- Edit Sketches
- Edit Feature Definition
- Edit Feature Parameters
- Deactivate Features
- Activate Features
- Changing the Sketch Support

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