



Creo Parametric 3.0 Basics - Part 1 (Getting Started with Creo Parametric 3.0 and Sketching)

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- Environments in Creo Parametric 3.0
- Parametric Modeling in Creo Parametric
- Associativity
- File Types in Creo Parametric
- Starting Creo Parametric 3.0
- User Interface
- Mouse Functions
- Background
- Shortcut Keys

Sketching

- Sketching in the Sketch environment
- Sketching Commands
- The Normal command
- Resolve Sketch
- Modify
- Constraints
- Turning ON/OFF Dimensions and Constraints
- Construction Mode
- The Circular Trim command
- The Circular command
- The Chamfer Trim command
- The Delete Segment command
- The Divide command
- The Corner command
- The Mirror command
- The Rotate Resize command
- The Offset command
- The Thicken command
- Palette



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