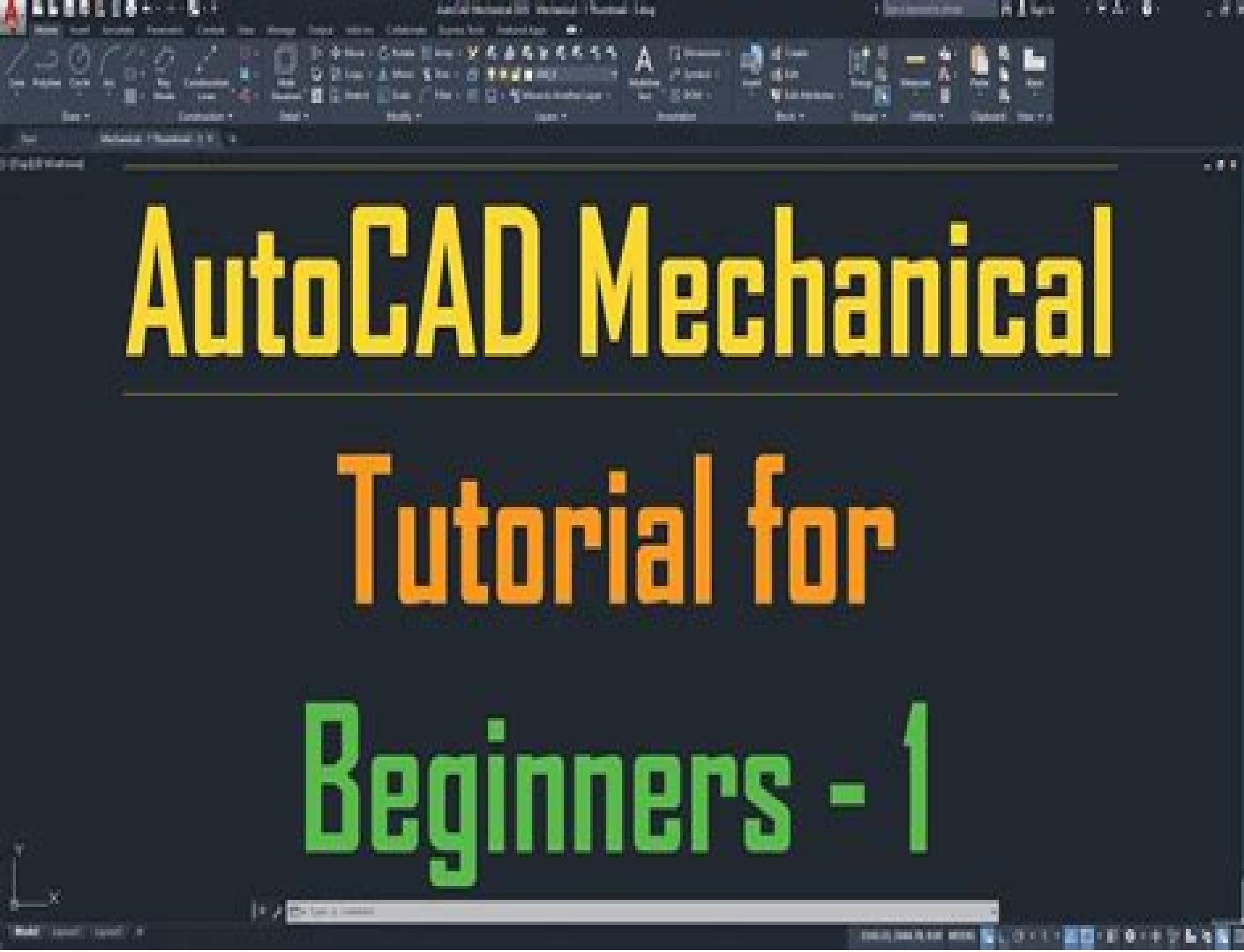




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DOS and Windows AME Provides a foundation for Release 13 solid modeling *Solidworks 2007 Tutorial with Multimedia CD* David C. Planchard, Marie P. Planchard, 2007 SolidWorks 2007 Tutorial was written to assist students designers engineers and professionals The book provides an introduction to the user interface menus toolbars concepts and modeling techniques of SolidWorks to create parts assemblies and drawings Follow the step by step instructions and develop multiple assemblies that combine over 80 extruded machined parts and components Formulate the skills to create modify and edit sketches and solid features Learn the techniques to reuse features parts and assemblies through symmetry patterns copied components design tables and configurations Review 2 hours of flash movie files that follow the steps in the book Desired outcomes and usage competencies are listed for each project Know your objective up front Follow the steps in Project 1 through Project 4 to achieve your design goals Work between multiple documents features commands and custom properties that represent how engineers and designers utilize SolidWorks in industry Table of Contents Introduction 1 Linkage Assembly 2 Front Support Assembly 3 Fundamentals of Drawing 4 Pneumatic Test Module Assembly Appendix Index Civil 3D 2007 Fundamentals Ascent -. Center For Technical Knowledge, 2015-01-09 Understand the full assembly functionality of Creo Parametric 2.0 while concentrating on techniques that maximize large assembly management capabilities as well as an introduction to Top Down Design This is a hands on learning guide with a substantial amount of time dedicated to exercises Topics include Advanced Component Placement Top Down Design Managing External References Assembly Management Skeleton Models Packaging Assembly Duplication Tools Assembly Family Tables Display Styles Layers and Suppression Restructure Creating Parts and Features in an Assembly Controlling External References Merge and Cut Out Intersections Copy Geometry Features Inheritance Features Simplified Representations Assembly Relations Interchange Assemblies appendix Prerequisites It is recommended to complete the following or have the equivalent Creo Parametric experience Creo Parametric 2.0 Introduction to Solid Modeling Part 1 Creo Parametric 2.0 Introduction to Solid Modeling Part 2 Creo Parametric Core Update Wildfire 4.0 to Creo Parametric 2.0 Please note that this learning guide uses commercial practice files which may not be compatible with the Student Edition of Creo Parametric

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