

Designing With Creo Parametric 2

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as an experienced user in the basics of creo parametric 3 0 the creo parametric 3 0 advanced part design student guide enables you to become more productive by extending your modeling abilities with advanced functionality and techniques this extensive hands on student guide contains numerous labs and practices to give you practical experience that will improve your job performance topics covered creo parametric fundamentals and interface advanced datum features variable section and helical sweeps blends and swept blends designing with rounds advanced round functionality drafts basic surface design part family tables advanced patterns and user defined features udfs date sharing view manager automation appendix prerequisites creo parametric 3 0 introduction to solid modeling or equivalent creo parametric experience

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black and white version of creo parametric 4 0 part 2 lessons 13 22 includes a complete set of lectures available on line through youtube for lessons and projects

note to complete this course creo parametric 2 0 introduction to solid modeling part 2 is required learn the process of

designing models with creo parametric 2 0 from 2d sketching through to solid part modeling assembly creation and drawing production gain an understanding of the design philosophy of creo parametric 2 0 through this extensive hands on course with numerous practice exercises it is expected that all new users of creo parametric 2 0 will require this course topics include creo parametric fundamentals and interface principles behind design intent manipulating a model creo parametric file management part creation and modification sketching and creating geometry sketcher mode functionality sketching and dimensioning datum features duplication techniques patterns mirroring creating relations to capture design intent creo parametric customization design documentation and detailing feature management sweeps and blends assembly creation and manipulation parent child relationships in creo parametric models model analysis feature failure resolution effective modeling techniques prerequisites experience in mechanical design and drawing production is recommended

this training guide enables you to use your introductory modeling skills to create sheet metal models including wall bends notches and form features on completion of this course the student will have acquired the skills to confidently manipulate sheet metal geometry adjust bend developed lengths and convert solid parts course topics the sheet metal environment primary and secondary walls bend relief corner relief regular unbends back bends and cuts notches and punches bend features unbending complex geometry sheet metal forms documenting a sheet metal part converting solid parts sheet metal setup investigating a sheet metal part prerequisites it is recommended to complete the following courses 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

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	know how	solidworks	solidworks	org	tw	forum php

this class is designed to improve your efficiency with creo parametric 2 0 by exploring and practicing the latest functionality in creo parametric 2 0 this course focuses on the enhancements within the core modules of creo parametric 2 0 and is ideal for those users updating from wildfire 4 0 to creo parametric 2 0 topics include general enhancements sketcher enhancements part enhancements advance part enhancements assembly enhancements drawing enhancements sheet metal enhancements prerequisites pro engineer wildfire introduction to solid modeling i ii or equivalent wildfire 4 0 experience

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