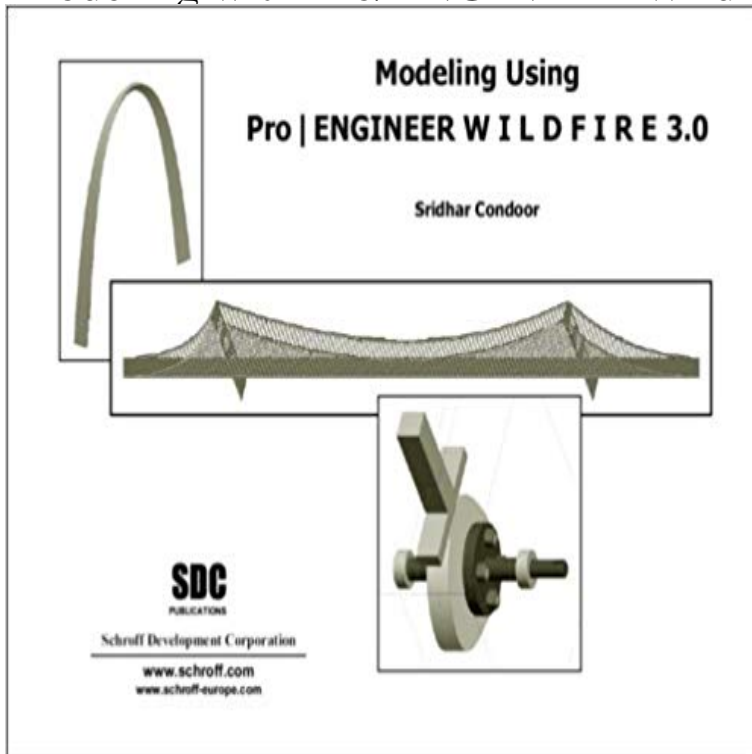


# Modeling with Pro/ENGINEER Wildfire 3.0



This book synergistically integrates the design process with the specific commands and procedures of Pro ENGINEER through a unique presentation scheme. Users are first provided with the design information about the part or assembly and its design intent. Then, they see the sequence of steps involved in modeling the part/assembly. Detailed instructions are provided in a four-column presentation showing goals, steps and commands. The consistent approach is supplemented by many illustrations on each page. Each chapter adds new information while reinforcing key concepts.

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