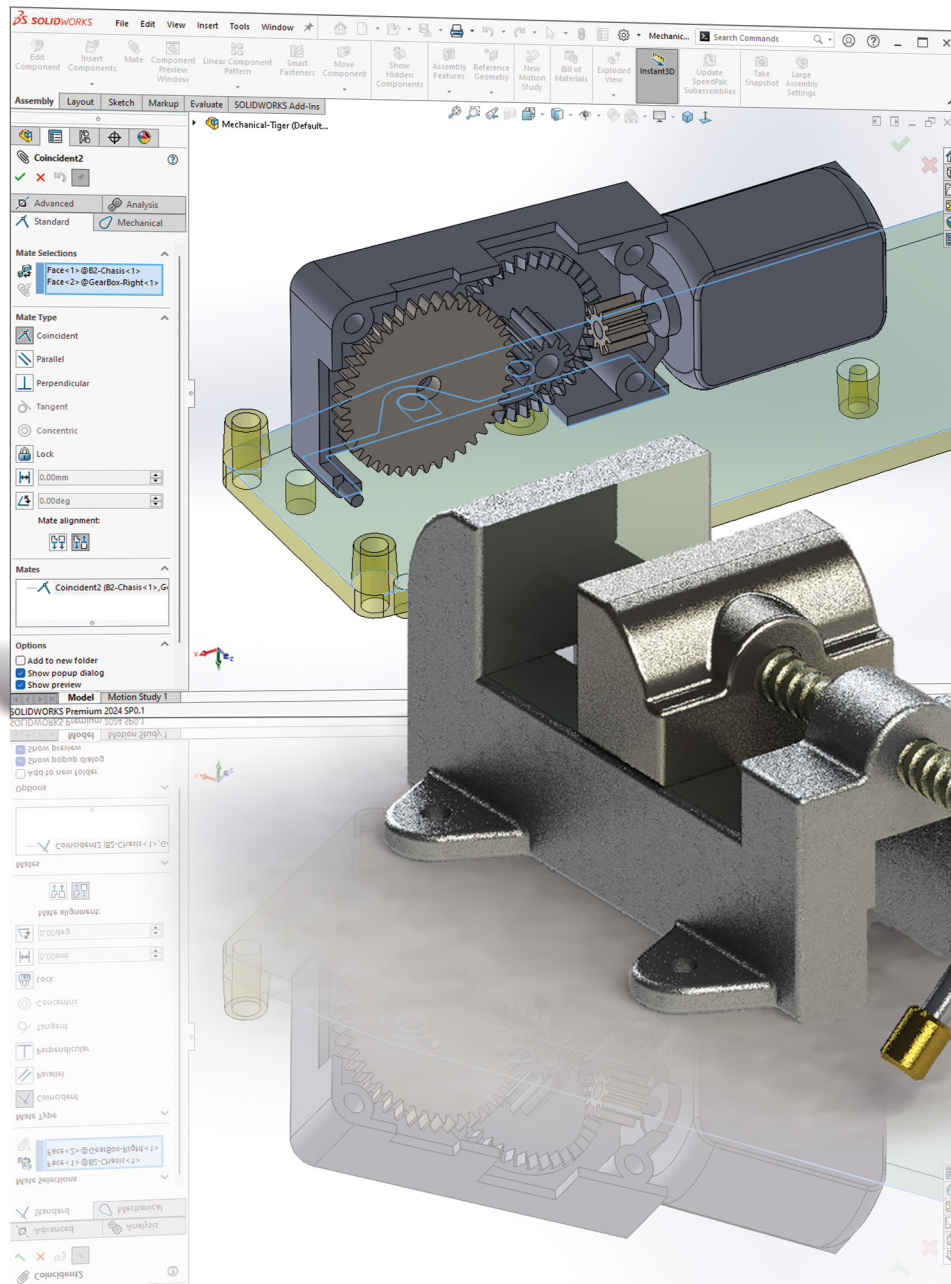


# Parametric Modeling with SOLIDWORKS® 2024

Covers materials found on the CSWA exam



Randy H. Shih  
Paul J. Schilling

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