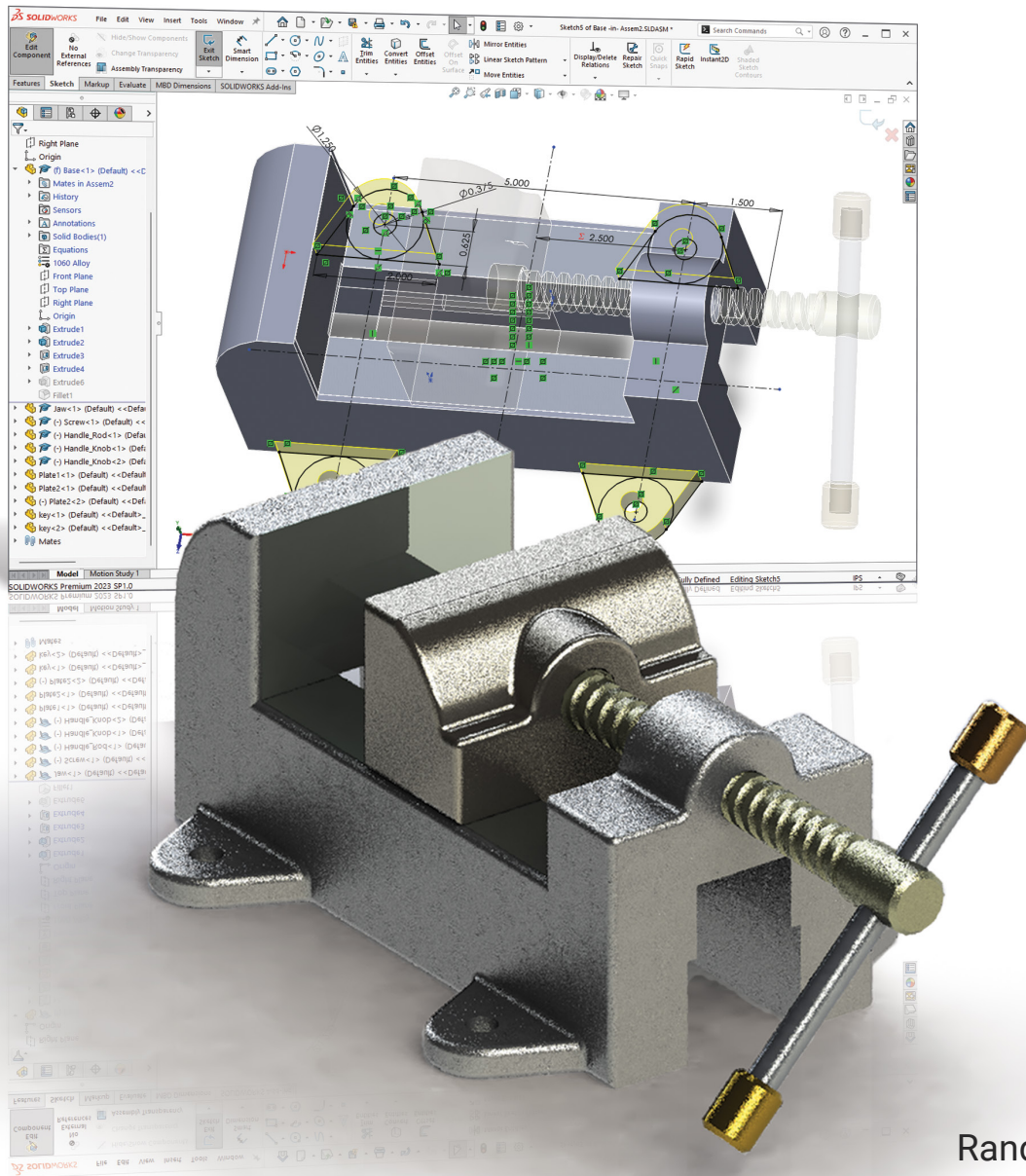


Parametric Modeling with **SOLIDWORKS**® 2023

Covers materials found on the CSWA exam



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