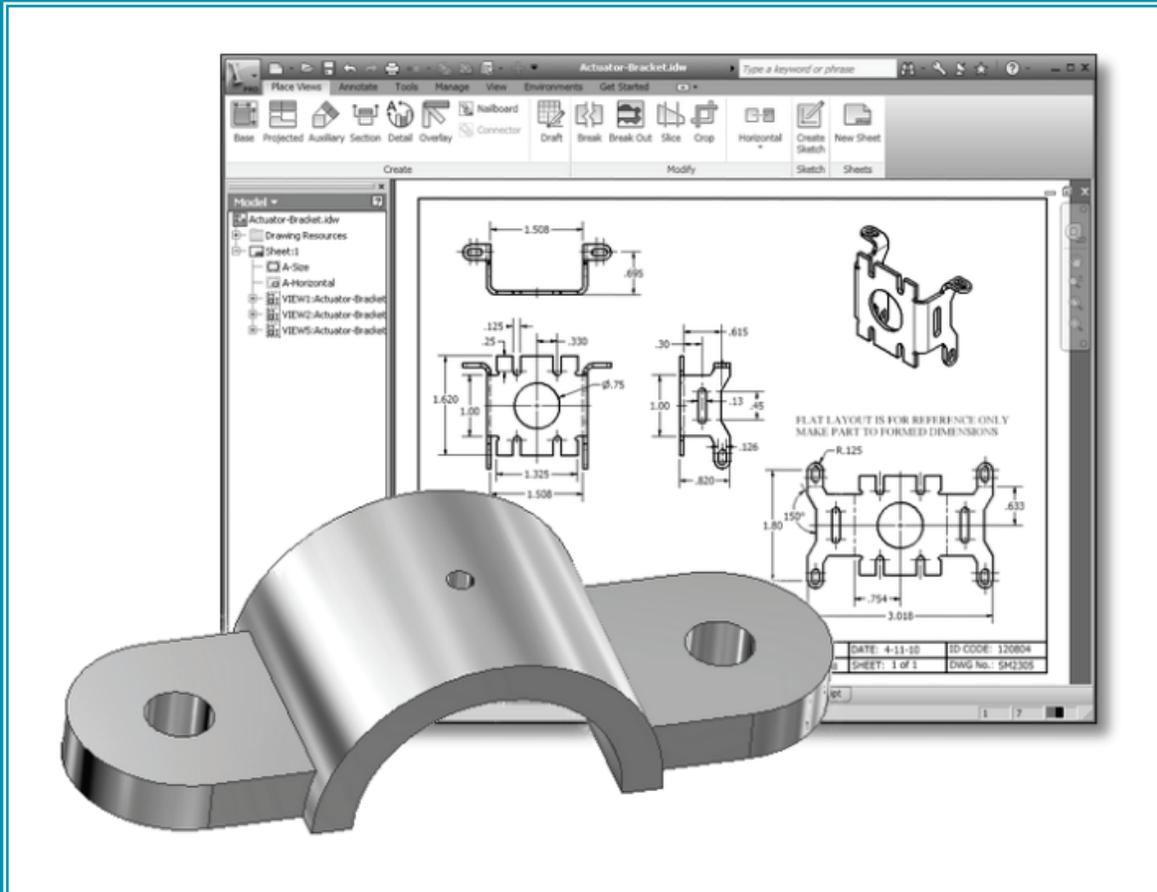


Parametric Modeling with Autodesk Inventor® 2011



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Autodesk Certification Program
The tutorials prepare students for
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