



# Machining Solid Edge Parts in NX CAM

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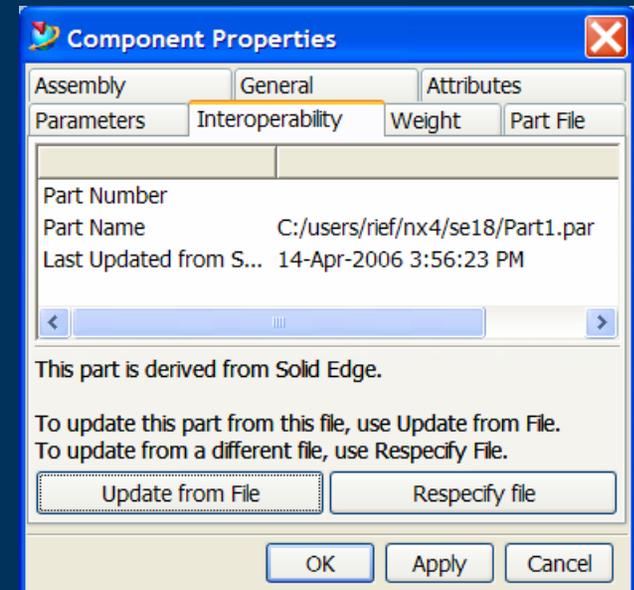




# Interoperability Basics



- ▶ NX can open a Solid Edge document
  - ▶ Part (.par) or Assembly (.asm)
  - ▶ Option to include PMI data – hole features
- ▶ Updating triggered through the Assembly Navigator
- ▶ NX part file references SE document
  - ▶ Part1.par ← Part1\_par.prt

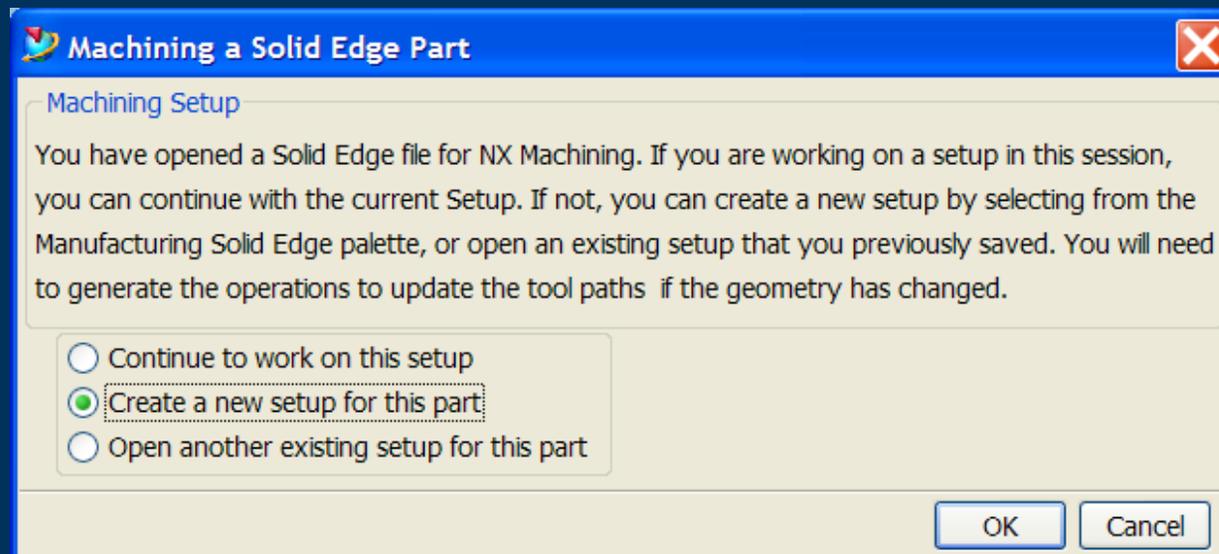
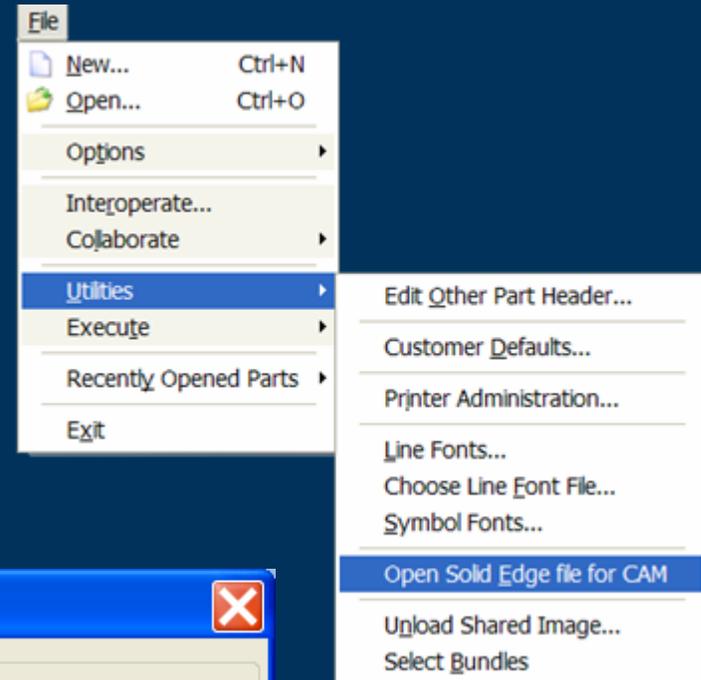




# NX 4.0 Interaction



- ▶ Two ways to activate
  - ▶ NX File → Utilities → Open Solid Edge File for CAM (roles with full menus, or customize)
  - ▶ SE Application → NX Machining
- ▶ Welcome dialog

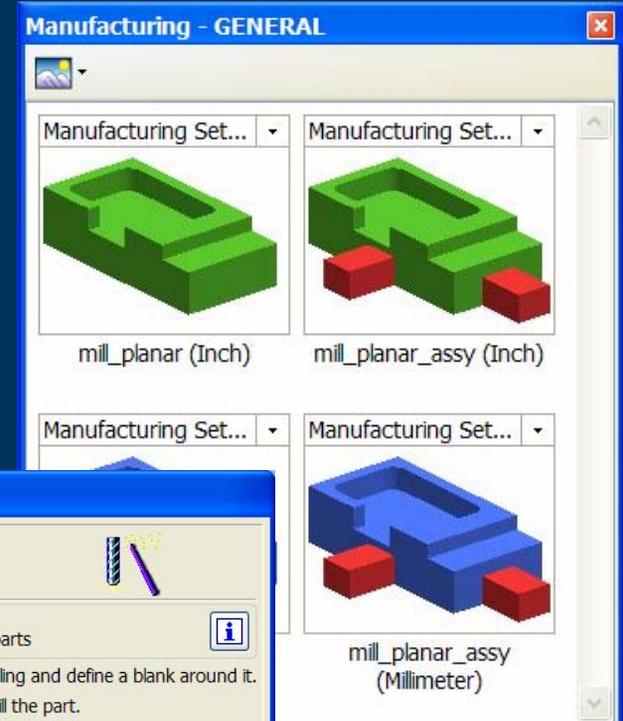




# NX 4.0 Interaction



- ▶ Setup Palette
  - ▶ Easiest way to start a setup assembly
  - ▶ Add Inch, Metric, or General
- ▶ Milling Quick Start Wizard
  - ▶ Selects Part





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