



Creo™ Parametric (formerly Pro/E) software interface

ProNest® optional module

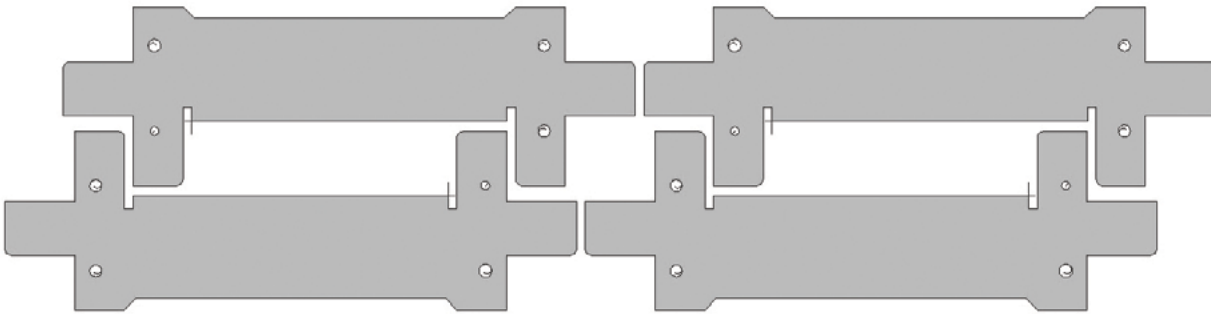
Benefits:

- Easier file exchange
- Faster programming

The Creo Parametric (formerly Pro/E) Software Interface module allows you to export Creo profiles, assemblies, and BOM information to *.dxf files that can then be imported to ProNest. This module installs directly into Creo Parametric providing menu options for Export, FlatCheck, BOM, etc.



No intermediate 2D file creation step is needed and therefore you'll have no concerns over managing revisions. Once imported to ProNest, you'll be able to work with these parts just like any others.



Also ask about Creo Parametric assembly export (sold separately).

Here is what people are saying about the Creo Parametric software interface:

"We needed a quick way to save our Creo parts as DWG files and this module works great. You just click Export and hit Enter and then you have your part. It can't get any easier. I'm not sure how we would get along without it."

- John Degnan, Universal Acoustic & Emission Technologies

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