

Pro/ENGINEER® Interface for JT

Frequently Asked Questions

What is Pro/ENGINEER Interface for JT?

Pro/ENGINEER Interface for JT is a new import and export processor for Pro/ENGINEER Wildfire® 4.0 designed to provide interoperability support to the JT file format. JT is a visualization format developed by Unigraphics and may be used to visualize and share 3D design data (parts and assemblies) between Unigraphics and Pro/ENGINEER users.

What does Pro/ENGINEER Interface for JT support?

Pro/ENGINEER Interface for JT is designed to translate structure, geometry and visualization information contained within JT format files. This interface supports import and export of:

- Product Structure
- Exact boundary representation data and/or facet data
- Body Colors

JT format does not support datum features or datum curves. JT data can be used within Pro/ENGINEER for visualization, measurements, and space claim.

Do I need to purchase Pro/ENGINEER Interface for JT?

Yes. Pro/ENGINEER Interface for JT is a separately licensed module for Pro/ENGINEER Wildfire 4.0 and later. There are two license types – one license for the individual desktop user and a separate license for large-volume, unattended processing via, for example, Distributed Pro/BATCH.

How do I install Pro/ENGINEER Interface for JT?

Pro/ENGINEER Interface for JT is distributed on the Pro/ENGINEER installation CDs. It is an optional component during the Pro/ENGINEER installation and is not installed by default. Administrators must choose to include the Pro/ENGINEER Interface for JT during Pro/ENGINEER installation.

How do I know if the Pro/ENGINEER Interface for JT was installed properly?

If properly installed and licensed, the JT option should appear in the Pro/ENGINEER File Open or Save a Copy dialog boxes as option in the file type drop-down list.

Can I evaluate Pro/ENGINEER Interface for JT?

Yes. PTC offers a time-limited evaluation license for Pro/ENGINEER Interface for JT free of charge. Your PTC sales representative can help you obtain an evaluation license for Pro/ENGINEER Interface for JT.

Can I use Pro/ENGINEER Interface for JT with Pro/ENGINEER Distributed Pro/BATCH?

Yes. Pro/ENGINEER Distributed Pro/BATCH introduces support for import and export of JT data with Pro/ENGINEER Wildfire 4.0 M010* and later.

Are there any optional settings for use with Pro/ENGINEER Interface for JT?

Yes. The JT format supports import and export of both exact (boundary representation) and tessellated (facet) geometry. This can be changed independently for import and export using the config.pro options `intf3d_in_import_as_facets` and `intf3d_out_export_as_facets` respectively with the optional values `default*`, `brep_only`, `facets_only`, `brep_and_facets`. The “default*” setting for the JT related config.pro options includes facet data only.

*** The timing of any product release, including any features or functionality, is subject to change at PTC’s discretion.**