

# Interface for Rhino in Pro/ENGINEER®

## Frequently Asked Questions

### **What is Interface for Rhino in Pro/ENGINEER?**

Interface for Rhino in Pro/ENGINEER is a new data exchange processor for Pro/ENGINEER Wildfire® 3.0 and later. It is designed to provide import-only support directly from Rhinoceros versions 1.0 through 4.0 files.

### **Why would I use Interface for Rhino in Pro/ENGINEER?**

This processor provides direct import of native Rhino files into Pro/ENGINEER. Eliminating an intermediate format transfer like IGES or STEP improves import quality. Rhinoceros is not needed for data exchange.

### **Do I need to purchase Interface for Rhino in Pro/ENGINEER?**

No. Interface for Rhino in Pro/ENGINEER is freely available to customers with active maintenance agreements.

### **What platforms are supported by Interface for Rhino in Pro/ENGINEER?**

Interface for Rhino is currently supported on Pro/ENGINEER supported Windows platforms (32 and 64 bit).

### **What entities does Interface for Rhino in Pro/ENGINEER support?**

Interface for Rhino in Pro/ENGINEER imports facet, exact, and curve data assigned to visible layers. Related color information is also imported based on the active color in the Rhinoceros files. Geometry import supports facet and exact data as separate facet and import features respectively within a single model.

### **Can I import my Rhinoceros file as a Pro/ENGINEER assembly?**

Yes. Rhinoceros is a part modeler but Rhinoceros users can create separate bodies and assign them to layers which represent individual assembly components. Interface for Rhino in Pro/ENGINEER interprets geometry (exact or facet) assigned to Rhinoceros layers as unique components within a flat assembly structure.

### **Is there a minimum supported build of Pro/ENGINEER Wildfire 3.0 required?**

The Interface for Rhino in Pro/ENGINEER works with Pro/ENGINEER Wildfire 3.0 (M140 or later\*). To ensure the best results, it is recommended to upgrade to the latest build of Pro/ENGINEER Wildfire 3.0.

### **How do I get the Interface for Rhino in Pro/ENGINEER?**

Interface for Rhino in Pro/ENGINEER is a GRANITE plug-in (“gPlug”) data exchange processor available as a [free download](#) from the PTC web site to Pro/ENGINEER Wildfire 3.0 customers with active maintenance agreements. The Rhinoceros gPlug is included on the Pro/ENGINEER Wildfire 4.0 (M010 and later) installation CDs and installed with Pro/ENGINEER Wildfire 4.0 and later by default.\*

### **How do I install the Interface for Rhino in Pro/ENGINEER Wildfire 3.0?**

Place the plug-in in the default Pro/ENGINEER plug-in directory, or put the gplug in a commonly accessible location and point to it with the gplug\_dir config.pro option. Please see the installation instructions available online for details.

### **How do I know if the Interface for Rhino in Pro/ENGINEER was installed properly?**

In Pro/ENGINEER, select File > Open > Type and look for “Rhinoceros (\*.3dm)”.

### **Can I use Interface for Rhino in Pro/ENGINEER with Pro/ENGINEER Distributed Pro/BATCH?**

Pro/ENGINEER Distributed Pro/BATCH does not currently support import of data from Rhinoceros files. Pro/ENGINEER Distributed Pro/BATCH support is being considered for a future release of Pro/ENGINEER.\*

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