This is a PDF version of Article CS124308 and may be out of date. For the latest version click here (https://www.ptc.com/en/support/article?n=CS124308&language=en&posno=1&q=master style&source=search)

Article - CS124308

Is it possible to disable the default style state from being retrieved automatically in Pro/ENGINEER and Creo Parametric

Created: 18-Apr-2013 | Modified: 07-Oct-2016

Applies To

- Pro/ENGINEER and Creo Elements/Pro Wildfire to Wildfire 5.0
- Creo Parametric 1.0 to 3.0

Description

- Is it possible to disable the **default style** state from being retrieved automatically ?
- Unable to retrieve assembly as **master style** by default
- Why do some models open in the **default style** and some others in the **master style** ?

Resolution

- Works to product specification for Pro/ENGINEER and Creo Parametric
- If the **default style** has been modified the model will be retrieved with **default style**, otherwise it will open with **master style**
- Workaround :
  - Remove customized definition of **default style**
  - Define wanted style with another name