PTC®

TS Bulletins

Unexpected exit when freeing memory though Erase Not Displayed (or other memory release) in Creo Parametric 2.0 M180 to M230 and Creo Parametric 3.0 M060 to M110

- Cause
  - Memory corruption issue
  - Only **some** models are affected
    - This issue was introduced in Creo 2.0 M180 and Creo 3.0 M060
    - Affected datecode ranges are **Creo 2.0 M180-M230** and **Creo 3.0 M060-M110**
    - For **some** models, if they are retrieved in the affected datecode range after being saved in a later/newer datecode, this issue **may** be reproduced (for example, avoid opening data in Creo 3.0 M100 if the data was previously created or opened and saved in Creo 3.0 M110, M120, etc.)
  - Generally, it is OK to work across different datecodes of the same release
    - This is an isolated issue, but to avoid it PTC suggests avoiding working in these datecode ranges if data has been created or modified in a future datecode
  - This issue can occur in a linked session with Windchill PDMLink and/or a Creo View Adapter worker session

- Resolution
  - Reported to R&D as SPR 6150170
  - Corrected in Creo Parametric 2.0 M240 and 3.0 M120
    - If you have models affected by this issue, upgrade all clients and workers to Creo Parametric 2.0 M240 or 3.0 M120
      - Once you upgrade to one of these datecodes, **do not** open the data again **in the affected datecode ranges**
    - Since this issue is only potentially reproduced by going to an earlier/older datecode in the affected ranges, you can also safely use the latest datecode in which data was saved
      - For example, if the issue is reproduced in Creo 3.0 M100 because the data was saved in M110, all clients/workers could adopt M110 to avoid the issue
      - Because this issue is dependent on the calendar build date of the datecode, not just the release, avoid moving from Creo 2.0 M220 to Creo 3.0 M090 as this issue has the potential to occur (since Creo 3.0 M090 was built before Creo 2.0 M220)
        - In general, follow the product calendar to avoid updating from Creo 2.0 to an "older" version of Creo 3.0, but you can also contact technical support for confirmation
    - If you are unable to upgrade to one of these datecodes, you can manually repair the data (see ***NOTE*** below)
      1. Obtain access to Creo 2.0 M170 or earlier or Creo 3.0 M050 or earlier
      2. Set config.pro option **save_objects** to **all**
         - If you see this issue with an assembly, it's not necessarily the assembly file causing the issue so this config.pro option will save every component
      3. **Retrieve** the dataset that reproduced the issue, **regenerate** it, and **save**
      4. This issue will no longer reproduce in Creo 2 M180 or later and Creo 3 M060 or later
      - ***NOTE***: after repairing the data manually, avoid retrieving and saving the data in a later build **and then** retrieving the data in an earlier build **in the affected ranges** (this could reintroduce the memory corruption)