

'INTERNAL ERROR' -OR- 'INTERNAL ERROR: MEMORY ACCESS VIOLATION' WHEN OPENING DRAWINGS/ASSEMBLIES IN NX12

Updated 2/19/19, Josh T.

There are at least two similar Internal Memory errors that have been encountered since the NX12 upgrade, but the causes and solutions are somewhat different.

ERROR TYPE 1: 'Internal Error'

SEE SOLUTION A OR B

This error generally prevents opening an assembly, not just a drawing. It can be triggered by both older and newer subassemblies/components that contain a 'MODEL' Reference Set. *NOTE: GTAC has confirmed that this issue is fixed in later NX releases.*

ERROR TYPE 2: 'Internal Error: Memory Access Violation'

SEE SOLUTION C

This error is more common and is generally encountered when attempting to open a drawing of an assembly, even when the associated assembly file opens just fine by itself. It seems to correlate with components that have un-optimized imported geometry (probably STEP files) and were generated prior to NX12. Unfortunately, parts that were not a problem one day (and therefore not addressed) may become a problem at a later date. However, parts that have been optimized per Solution C do not seem to have recurring problems themselves.

SOLUTIONS:

NOTE: In order to save time and to provide immediate feedback to the user, it is recommended that the following solutions be performed by opening the problematic file by itself in a second NX Manager session. The following solutions assume the user is running two concurrent NX Manager sessions.

IDENTIFY Problematic Subassemblies and/or Components (for BOTH error types):

1. Load the Assembly or Drawing with Assembly Load Options set to *Structure Only*.
Additional load options (other combinations may be okay; this is what I use):
 - a. For Assemblies: Partially Load - Lightweight Display / Interpart Data OFF.
 - b. For Drawings: Fully Load - Lightweight Display / Interpart Data OFF.
2. Expand the Assembly Navigator Tree and click on subassemblies one at a time to try to open them (or narrow down the problem more quickly by highlighting multiple subassemblies and attempting to open several at once).
3. If clicking a part triggers an error message, try Right-click > Open > Component.
4. If the component opens, repeat steps 2-3 for child and grandchild components.
5. If the component (or a child/grandchild component) does NOT open by Right-Click > Open > Component, you have located a problematic file that needs to be addressed.

SOLUTIONS FOR 'INTERNAL ERROR':

Solution A (Delete the Model Reference Set of a subassembly):

1. Identify the problematic file per the suggested workflow (above).
2. Open the component by itself (in a second session).
3. Go to Menu > Format > Reference Sets
4. Delete the Model Reference Set.
5. Check out and save the part.
6. In the original NX Manager session, click the problematic part in the Assembly Navigator to turn it on.

Solution B (Replace the Model Reference Set of an individual part):

1. Identify the problematic file per the suggested workflow (above).
 2. Open the component by itself (in a second session).
 3. Go to Menu > Format > Reference Sets
 4. Delete the Model Reference Set.
 5. Create new reference set named 'MODEL' and select the body of the part for inclusion.
 6. Check out and save the part.
 7. In the original NX Manager session, click the problematic part in the Assembly Navigator to turn it on.
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SOLUTIONS FOR 'INTERNAL ERROR; MEMORY ACCESS VIOLATION':

Solution C (Optimize a part consisting of imported geometry/STEP files):

1. Identify the problematic file per the suggested workflow (above).
2. Open the component by itself (in a second session).
3. Look at the Part Tree to verify that the part is made up of bodies only.
4. Optimize Face / Drag Select all the faces of the body(s).
5. Check out and save the part.
6. In the original NX Manager session, click the problematic part in the Assembly Navigator to turn it on.
7. If part does not turn on and still gives the error, replace the part's Model Reference Set (*see Solution B*) and then try to turn it on again in the original session.

Solution D (Check-out/Save only -- NOT RECOMMENDED unless there is a compelling reason to avoid using Optimize Face in Solution C):

1. Identify the problematic file per the suggested workflow (above).
2. Open the component by itself (in a second session).
3. Check out and save the part (this merely updates the file to NX12 format).
4. In the original NX Manager session, click the problematic part in the Assembly Navigator to turn it on.
5. NOTE: This solution may not prevent the error from recurring in a future NX session since it does not address potentially problematic solid geometry.