

Criteria for Successful Drawing Version Up

There are three critical factors customers should be aware of to ensure successful drawing version up.

1. [Ensuring drawings are ready for compatible version up](#)
2. [The version up process using refile](#)
3. [Understanding acceptable differences](#)

Ensuring Drawings are Ready

Drawings must pass the following criteria in order to ensure compatible version up:

1. The model geometry must version up without issue.
2. The model geometry passes validation by Examine Geometry.
3. All drawing views have adequate display tolerances.
4. All drawings are saved with up-to-date views.
5. Views with interfering solids are saved with the interfering solid option on.
6. Drawing files are in sync with all sub assemblies and components.

The Version Up Process

For successful results with pre-NX (V19) parts, the version up should be performed using the refile_part program with the -drawing option and with the following environment variables set.

Variable	Value	Description
UGII_FIXUP_SHIFTING_DATUMS	1	Correction for datums shifting
UGII_DONT_CONVERT_WELD_SYM	ON	Does not convert DENSO specific welds (Only required by Denso customer)
UGII_FORCE_PRE_V17_THREAD_MIN_PITCH_TO_ZERO	1	Required for parts saved in versions after V16 that did not use the environment variable below.
UGII_FORCE_THREAD_MIN_PITCH_TO_ZERO	1	Preserves thread display

Drawing Refile Overview

Drawing Refile aids in the migration of legacy drawings to later versions of NX. Its purpose is to maintain the display of the legacy drawing *even after* the drawing is updated in a later version of NX.

What Refile Does

Refile updates each view and modifies objects to maintain their original appearance.

1) View boundary shape and size

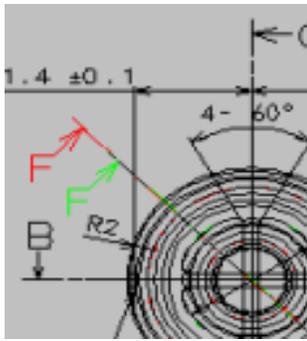


V18

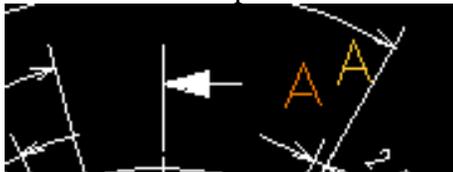


NX2

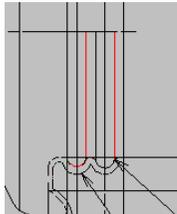
2) Section line arrow position



3) Section line label position



4) Virtual Intersections



5) Hidden line on tolerant edges

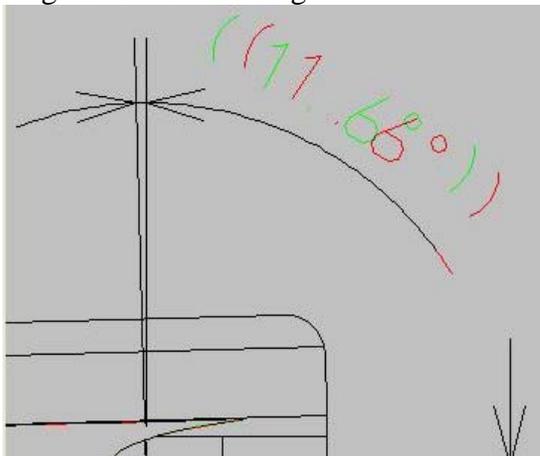
6) Annotation origin position relative to a view



7) View label position



8) Angular dimension origin



NOTE: There are exceptions for changes that do not impact engineering intent (otherwise known as Acceptable Differences). An example of an acceptable difference would be a shift in the pattern of a crosshatched area. See Acceptable Difference sections below.

When to use Drawing Refile?

You only need to use drawing refile on drawings that you plan to update in the new release. If you are only opening and reviewing a drawing without saving it in a new release, there is no need to run the refile program. The changes described above only occur after a view update.

Why did the changes occur?

In every release of NX, the code is enhanced with many customer fixes. These fixes do not take place until after a drawing view is updated. As a simple example, view borders are calculated to be tighter to the geometry in V18. This had the impact of shifting any view dependent annotation (view labels for example) or symbols (section line arrows and labels for example). These shifts are seen as changes in the appearance of the drawing and thus are unacceptable. In this particular example, refile recalculates the position of the annotations and symbols to be located in the same place relative to the new view boundary. Other fixes were made to virtual intersection curves and angular dimension location.

Some things to note with Refile

1. Once a part is saved in a later version using drawing refile, you do not need to rerun the part through drawing refile again.
2. Virtual intersection curves were improved. Thus in an updated view, you may see new virtual intersection curves appear that were not there or curves that did appear may have been incorrectly calculated. With Refile, the system will maintain any incorrectly calculated curve as a non-associated curve and put it in a group called "OLDVICURVES". Similarly, for new curves that did not appear before, the system will erase these curves and place them in a group called "NEWVICURVES". This way, the drawing view is preserved, but you have access to the correct curves if you so desire to use them.

How to use Refile

You can access the drawing refile capabilities by using:

- Natively: refile_part.exe -dwg -y -f <drawing part file name>
(see user documentation for other options).
- TCEng: when refiling, enable the "refile_drawings" option
(not available until NX3.0.3).
- Interactively: Set the following environment variables:
UGII_DRAFT_DRAWING_UPGRADE =1
UGII_SUPPRESS_VIEW_UPDATE=1
Legacy drawings will upgrade when the drawing part is loaded.
(not available until NX3.0.3).

Acceptable Drawing Differences

Acceptable drawing differences are those that do not change the interpretation or legibility of the drawing.

By interpretation, we mean that the drawing should not change in a way that may cause the part to be manufactured differently. Examples of these cases would be changes in dimension values and missing or additional annotation and geometry.

By legibility, we mean that the drawing should not change in a way that causes the drawing to no longer conform to standards or to the original look of the drawing. Examples of these cases are mostly limited to significant shifts of annotation and drafting objects.

Acceptable differences generally fall into two categories

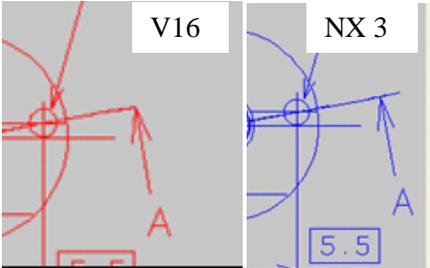
1. Problems or limitations existed in the earlier versions that caused drawings to be displayed contrary to drafting standards. In these cases we fix the problem or limitation and accept the difference so long as the interpretation and legibility of the drawing are not affected. See 1, 2, 5, 8, and 9 below for examples.
2. Minor appearance deviations are accepted if resolving them would cause too much complexity and hence instability and worsened performance within the drafting application. Examples of this are how dashed lines are rendered. By definition these appearance problems neither affect interpretation nor legibility and so we accept the difference. See 3, 4, and 6 below for examples.

Any other difference is treated seriously by UGS and is resolved as rapidly as possible.

The following section documents some of the differences that meet the above definition for acceptable differences. *This section may grow to include other examples in the future.*

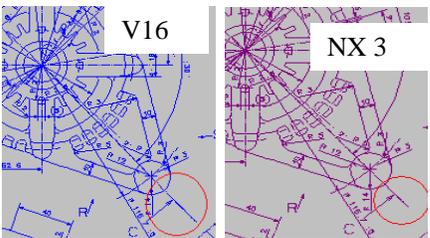
Examples of Acceptable Drawing Differences

1. Section Line Conforms to Standards in V18 and later



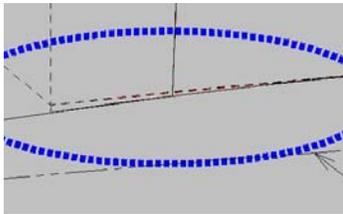
Description: An error occurred in V16 where the section line did not extend past the section line arrow. This is a problem in V16 that was corrected in V18 and later releases.

2. Manually Moved Section Line Arrow Issue



Description: An error occurred in V16 where if the section line arrow is manually moved by the user, it did not update the extension past the location of the arrow. The NX 3 correction now renders the extension past the section line arrow per the part setting.

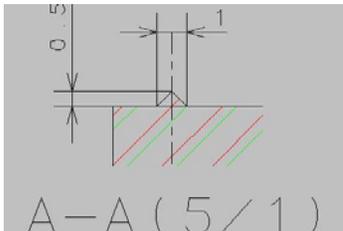
3. Dashes Do Not Match Exactly



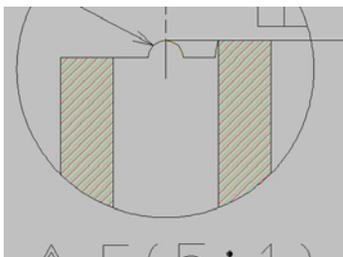
Description: The start and end of dashes may not coincide with those in the previous release.

As long as the size of the dashes does not change considerably and the meaning of the dashed line on the drawing does not change the lines interpretation on the drawing, there is no plan to address this difference.

4. Crosshatch Shift



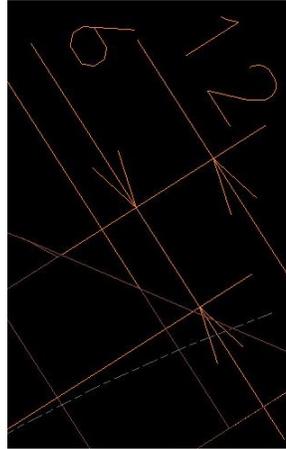
Description: The crosshatch lines shift in location only. There is no change in angle, spacing, font, and width. The shifted lines remain trimmed to the same crosshatch area.



5. Arrows Move to Outside



V16

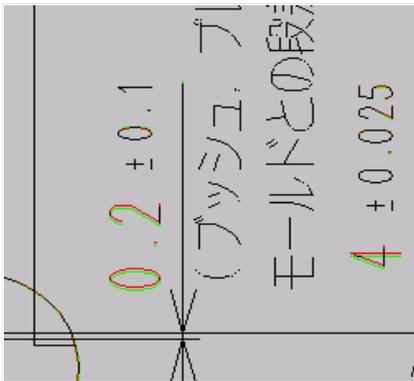


NX 3

Description: If you manually update the dimension in V16, the arrows will flip to the outside. In V16, we did not update the dimensions on view update unless there was a change in value. This behavior caused problems in that all dimensions may not be up to date after a view update. In NX 3, all dimensions are updated when a view is updated. So, the behavior that the arrows flip when the dimension is updated is consistent between V16 and NX 3. What is new in NX 3 is that all dimensions update on view update.

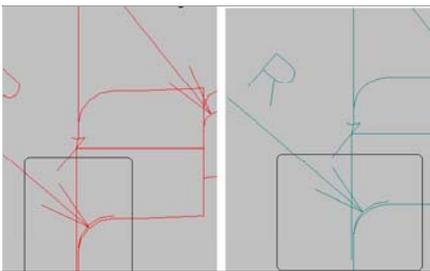
Moreover, the behavior in NX 3 makes the drawing more legible and behaves as expected.

6. Minor Text Shift



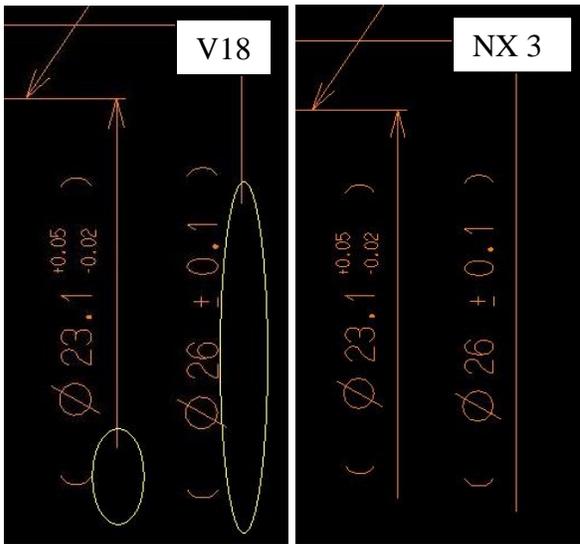
Description: Minor shift in annotation. No other changes in annotation color, font, and type occur.

7. Arrow Points to Wrong Curve



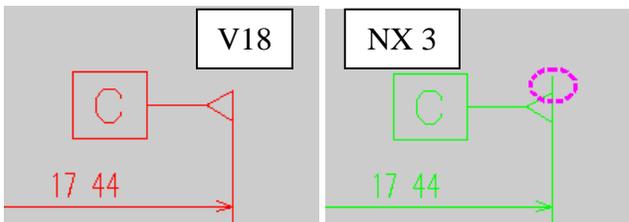
Description: In V16, the dimension pointed to the wrong curve. Thus the dimension value did not truly represent what it was pointing to. In NX 3, the system was corrected so the dimension points to the right curve. This is evident since the dimension value did not change.

8. Dimension Line Now Extends Under Entire Dimensions



Description: In prior releases, the dimension line did not extend under the entire dimension per standard specification. This is corrected in NX 3.

9. Datum Extension Line



Description: The extension line did not exceed pass the datum suction cup per standard specification. This is now corrected in NX 3.