

# NX Environment Variables

## From PLMWiki

The following are variables that can be set for NX as a system environment, startup script or customer defaults file. The more common customer defaults file are used the less these variables. Then these variables will be used for testing or debugging.

## NX 6 Added Variables

turn on / some of the legacy transformation options are not available in NX6.

- UGII\_ENABLE\_TRANSFORM\_LEGACY\_OPTIONS = 1

turn on the new NX6 Hole Table creation in drafting

- UGII\_DRAFTING\_HOLE\_TABLE=1

## NX 5 Added or Deprecated Variables

Enable the Class Selection Dialog Removes the floating dialog class selection and goes straight to the selection dialog

- NX\_FORCE\_CSD=1

Point to the directories with the template and pax files (File\New)

- UGII\_TEMPLATE\_DIR=<path>

Enable on extra Nx5.02 Replace Face Option

- UGII\_Simplify\_Surface=1

After Nx5.02 activate old Edit->Face functions

- UGII\_DMX\_NX502=1

don't cache WAVE specific information

- UGII\_DONT\_CACHE\_WAVE\_LINK\_DATA=1

Variable to Aid With "Out of Memory" Errors on Large Assemblies

- UGII\_CLOSE\_UNDO\_MODE=CHECK

## NX 4 Added or Deprecated Variables

Visually see the coverage of a Journaling when creating a Journal file

- UGII\_JOURNAL\_INDICATOR=1

Enable view dynamics under the function keys (F1 -> F4)

- UGII\_FKEY\_DYNAMICS=1

Required to load the Solid Edge V19 adapter when Solid Edge parts are opened in NX4.0.2

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- UGII\_USE\_SEV19=1

## NX 3 Added or Deprecated Variables

Store user customizations

- UGII\_LOCAL\_USER\_DEFAULTS=<path>\nx3\_user.dpv

Set Default Role, MTX file must be in the ugii\menus\roles\

- UGII\_DEFAULT\_ROLE=example.mtx

Location of the user.mtx file Store nx\_mru.txt file for folding of menus

- UGII\_USER\_PROFILE\_DIR=<path>

Save history.pax palette file in a specific location

- UGII\_HISTORY\_PALETTE\_DIR=<path>

Turn of folded menus and prevent recording actions

- UGII\_NO\_FOLDED\_MENUS=1

Default location for interactively created palettes

- UGII\_PALETTE\_WORK\_DIR=<path>

NX to perform an interactive drawing refile on part load

- UGII\_DRAFT\_DRAWING\_UPGRADE=1

Enable the Class Selection Dialog

- NX\_FORCE\_CSD=1

Disable splash on start of a NX session

- UGII\_DISABLE\_FLASH=YES

Enable the drawing compare (the hpgl output) function under file -> utilities ->

- UGII\_ALLOW\_DRAWING\_COMPARE=1

## Base Environment Variables

Not sure which version it was first released in but they have been in NX for a while.

Support for Multi Processor or Multi Core CPUs

- UGII\_SMP\_ENABLE=1

Mouse wheel (since NX2)

- UGII\_MOUSE\_WHEEL=1 default, roll back to enlarge
- UGII\_MOUSE\_WHEEL=2 roll forward to enlarge
- UGII\_MOUSE\_WHEEL=0 inactive mouse wheel, no zoom

Where should execute NX/Open application directory look by default

- UGII\_INITIAL\_UFUN\_DIR=

Allow for opening of Solid Edge Assemblies

- UGII\_SE\_INTEROP=1

All the expression editor to be viewed in Drafting

- UGII\_DRAFT\_EXPRESSIONS\_OK=1

Required for interoperability to function between Solid Edge and NX

- UGII\_MONIKER=1

NX Location

- UGII\_BASE\_DIR=C:\PROGRA~1\UGS\NX4~1.0\
- UGII\_ROOT\_DIR=%UGII\_BASE\_DIR%UGII\

Point to the application directory instead of custom\_dirs.dat

- UGII\_USER\_DIR=<path>

Set Custom Directory Location

- UGII\_CUSTOM\_DIRECTORY\_FILE=<path>\custom\_dirs.dat

set part family member directory

- UGII\_INITIAL\_FAMILY\_INSTANCE\_DIR=%folder%

Enable class selection on the graphics display dialog bars

- UGII\_CSD\_ON\_DIALOG\_BAR=1

Enable class selection on the selection toolbar

- UGII\_CSD\_ON\_TOOLBAR=1

Text file to map directories between windows and unix

- UGII\_DIRECTORY\_MAP\_FILE=<path to file>

Prevents the syslog from being removed from temp

- UGII\_KEEP\_SYSTEM\_LOG=1

File difference time tolerance

- UGII\_MODIFICATIONS\_TIME\_TOLERANCE=60

Prevents NX from defaulting to sketch on face when extruding Customer Defaults: Modeling->Extras->Sketch on Planar Faces

- UGII\_SECTION\_USE\_PLANAR\_FACE=1

Default location of your pattern files

- UGII\_PATDIR=<path>

Creates a new parts list from a template

- UGII\_DEFAULT\_PARTS\_LIST=<path>

Remove unsupported characters from text in drafting

- UGII\_DRAFT\_REMOVE\_UNSUPPORTED\_CONTROL\_CHARACTERS=TRUE

Enable symbolic thread performance improvements

- UGII\_DRAFT\_SUPPRESS\_THREAD\_TRUE\_HIDDEN\_LINE=1

Force Parts lists to Upper Case

- UGII\_FORCE\_PLIST\_AUTO\_TEXT\_TO\_UPPER\_CASE=1

Prevent Internal parts list from being used for Insert->Parts List

- UGII\_IGNORE\_INTERNAL\_PLIST=1

Delay View Updates preference can be saved with the part file, this overrides that setting

- UGII\_SUPPRESS\_VIEW\_UPDATE=1

Enable Surface Finish symbols in drafting

- UGII\_SURFACE\_FINISH=ON

Defining a user defined location for custom symbols

- UGII\_SYMBOL\_LIBRARY\_DIR=<path>

User Defined Directory for Table Templates defined by pax files

- UGII\_TABLE\_TEMPLATES=<path>

Allows user defined balloon symbols to be applied to assembly drawings

- UGII\_UPDATE\_ALL\_ID\_SYMBOLS\_WITH\_PLIST=1

Activate the weld symbols utility for drafting

- UGII\_WELD\_SYMBOL=ON

Converts promotions to WAVE geometry

- UGII\_CONVERT\_PROMOTION\_TO\_LINKED\_BODY=1

turn off preview generation for all component Previews will still be generated for the Work and Display part

- UGII\_DISABLE\_COMPONENT\_PREVIEWS=TRUE

Disable the generation of preview images under all circumstances

- UGII\_DISABLE\_PREVIEW=ON

Overrides customer defaults or the Save Options to always save trueshape data

- UGII\_FORCE\_TRUESHAPE\_SAVE=1

Reverts to old style if internal errors received when adding a component to an assembly

- UGII\_OLD\_STYLE\_STAGING\_VIEW=YES

disable part preview in Open Part file

- UGII\_SHOW\_PART\_PREVIEW=OFF

Allows JT assemblies to be opened as an assembly

- UGII\_JT\_ASSY\_STRUCTURE=1

auto update for parts lists will not be turned off in NX/Manager mode

- UGII\_UGMGR\_PLIST\_ALLOW\_AUTO\_UPDATE=1

point UG to the install path of Process Aid Wizard

- UGPROCESS\_AID=<path>

Enable Weld Assistant Required for weld assistant. Points UG to the install directory for weld assistant. E.g:  
D:\Program Files\EDS\Unigraphics NX\UGWELD

- UGWELD\_DIR=<path>

## Helpfull Part Attributes

-->File->Preferences

If attribute exists on a component then component is excluded by all parts list

- PLIST\_IGNORE\_MEMBER=1

If attribute exists on a component then that component and all of its children is excluded

- PLIST\_IGNORE\_SUBASSEMBLY=1

If attribute exists on a component will not intersected and shown with no crosshatching in drafting section view

- SECTION-COMPONENT=NO

Retrieved from "[http://plmwiki.com/NX\\_Environment\\_Variables](http://plmwiki.com/NX_Environment_Variables)"

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