

# *What's New in NX 6.0.5*

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## Chapter

# *1 Introduction to What's New*

The What's New Guide briefly summarizes the new features in each release.

This guide highlights what each function does, why it should be used, and where it can be found in the user interface. This guide also conveys the benefit of each new capability.



## Chapter

# 2 *PCB.xchange*

### Use existing part attribute as default for component number

#### What is it?

You can now define a user-specified component number attribute from which PCB.xchange reads the component number instead of reading it from the *MAYA\_PCB\_COMP\_NUMBER* variable.

To activate this feature, set the following variable in the *pcb\_xug\_model.ini* file as in the following example:

```
NxComponentNumberSecondaryAttName = DB_PART_NO
```


#### Note

*DB\_PART\_NO* is an example. You can specify any attribute variable.

#### Why should I use it?

Use this feature when, for example, you create multiple components in PCB.xchange from the Teamcenter database where *DB\_PART\_NO* is already defined.

#### Where do I find it?

Application	PCB.xchange
Prerequisite	Modify the <i>pcb_xug_model.ini</i> file before launching NX.
Toolbar	<b>PCB.xchange toolbar</b> ® <b>Import ECAD Model</b> 
Menu	<b>PCB.xchange</b> ® <b>Import ECAD Model</b>

The *pcb\_xug\_model.ini* file can be found in the following locations:

- NX installation directory
- Location specified by *MAYA\_PCB\_DIR*
- Network Location specified by *MAYA\_PCB\_ENV\_DIR*

## Exporting restriction area to IDF enhancement

### What is it?

When you export the PCB restriction areas to IDF format, you can now specify the ECAD layer to which the area will be written.

To define the restriction area options for the IDF format, customize the following variables in the *pcb\_xug.ini* file as follows:

```
KeepInAreaType = KIAreaType, ECADLayer

KeepOutAreaType = KOAreaType, ECADLayer

OtherAreaType = OTHERAreaType, ECADLayer
```

The first keys, `KIAreaType`, `KOAreaType`, and `OTHERAreaType`, are the restriction area types. These types did not change in this NX version. See [Variables in the pcb\\_xug.ini file](#) in the NX *PCB.xchange Help* for more information.

The second key, `ECADLayer`, can now be set when you export to IDF format. The following `ECADLayer` are recognized by the IDF format:

- `other_outline` — Exports your restricted area as an Other Other Area.
- `route_outline` — Exports your restricted area as a Routing Keep-in Area.
- `place_outline` — Exports your restricted area as a Placement Keep-in Area.
- `route_keepout` — Exports your restricted area as a Routing Keep-out Area.
- `via_keepout` — Exports your restricted area as a Via Keep-out Area.
- `place_keepout` — Exports your restricted area as a Placement Keep-out Area.
- `place_region` — Exports your restricted area as a Placement Group Keep-in Area.


### Why should I use it?

Specify the ECAD layer for the restriction area types when you want, for example, to model areas where to place components as keep-in areas and export them to IDF format as keep-out areas.

### Where do I find it?

Application	PCB.xchange
-------------	-------------



Prerequisite	Modify the <i>pcb_ug.ini</i> file before launching NX.
Toolbar	<b>PCB.xchange toolbar</b> ® <b>Export ECAD Model</b> 
Menu	<b>PCB.xchange</b> ® <b>Export ECAD Model</b>

The *pcb\_ug.ini* file can be found in the following locations:

- NX installation directory
- Location specified by MAYA\_PCB\_DIR
- Network Location specified by MAYA\_PCB\_ENV\_DIR

## Automatically use component's reference set

### What is it?

When you import an ECAD model in PCB.xchange, PCB.xchange can load different instances of a component part using reference sets as the component number.

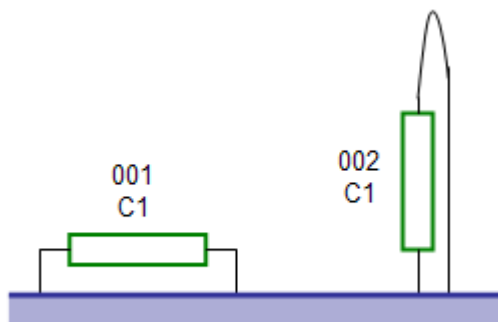
These component instances are saved in the same NX Part. Each instance is a solid body with a specific reference set.

For each instance of the component part, set the **Part Number** in the **Component Attributes** dialog box to its specific reference set name.

To activate this feature set the following variable in the *pcb\_ug\_model.ini* file as follows:

```
NXWriteCompRefSetAsCompNumber = Yes
```

For example, in this graphic, the same component C1 is placed once horizontally and once vertically on the PC board. The horizontally placed component has a reference set named **001**, and the vertically placed component has a reference set named **002**. Both are saved in *C1.prt*.



In the **Component Attributes** dialog box, set **Part Number** to:

- 001 for component C1-001.
- 002 for component C1-002.

After you export this PCA to ECAD and then re-import it, PCB.xchange loads the appropriate instance of the component.

**Note**


When a component part has more than one instance and you do not specify the reference set for each instance, PCB.xchange loads all instances of that component.

**Why should I use it?**


Use this feature when you have some components that have the same package name, but are oriented differently or have different variants or shapes. For example, you may want to use the simplified version of your component for a thermal analysis, and the detailed version for evaluating placement interference.

**Where do I find it?**

To modify a component attribute

Application	PCB.xchange
Prerequisite	Modify the <i>pcb_xug_model.ini</i> file before launching NX.
Toolbar	<b>PCB.xchange toolbar</b> ® <b>Component Attributes</b> 
Menu	<b>PCB.xchange</b> ® <b>Edit Attributes</b> ® <b>Component</b>
Location in dialog box	<b>Component Attributes</b> ® <b>Part Number</b>


To export an ECAD model

Application	PCB.xchange
Prerequisite	Modify the <i>pcb_xug_model.ini</i> file before launching NX.
Toolbar	<b>PCB.xchange toolbar</b> ® <b>Export ECAD Model</b> 
Menu	<b>PCB.xchange</b> ® <b>Export ECAD Model</b>

To import an ECAD model

Application	PCB.xchange
Prerequisite	Modify the <i>pcb_xug_model.ini</i> file before launching NX.

---

Toolbar	<b>PCB.xchange toolbar</b> ® <b>Import ECAD Model</b> 
Menu	<b>PCB.xchange</b> ® <b>Import ECAD Model</b>

The *pcbx\_ug\_model.ini* file can be found in the following locations:

- NX installation directory
- Location specified by MAYA\_PCB\_DIR
- Network Location specified by MAYA\_PCB\_ENV\_DIR



## Chapter

# 3 *Advanced Simulation*

## Supported solver versions

### What is it?

For each released version of NX, the following tables list the supported solver versions for import, export, and the post-processing of results. Note:

- The version listed in the “Import ASCII” and “Import Binary” rows is the solver version that was generally available when the NX version was released. In general, the import of the solver ASCII and binary files should be upwards compatible. Therefore, you should be able to import them into the most recent version of NX. However, in general:
  - ASCII files are backwards compatible for import into NX. If you import an ASCII file from a newer version of the solver than is officially supported, the software simply ignores any new fields/options that aren’t supported in the current NX release.
  - Binary files are not backwards compatible. For example, you can import a binary file created by NX Nastran 5.0 into NX 6.0.2 , but you might not be able to import a binary file created by NX Nastran 6.1 into NX 5.
- The version listed in the “Export ASCII” rows is the solver version that was available when the NX version was tested. In general, the exported solver input file is upwards compatible for that solver. Backwards compatibility is not guaranteed. For NX Nastran, the **Model Setup Check** function in Advanced Simulation tries to flag potential version incompatibility issues.
- The version listed in the “Post-processing Results” rows is the version of the solver results that was tested in the listed NX version. In general, results from earlier solver versions are also supported.

**NX7 releases**

<b>Solver</b>	<b>File Type</b>	<b>NX 7</b>	<b>NX 7.0.1</b>
<b>NX Nastran</b>	Import ASCII (.dat)	6.1	7.0
	Import Binary (.op2)	6.1	7.0
	Export ASCII (.dat)	6.1	7.0
	Post-processing of Results	6.1	7.0
<b>MSC Nastran</b>	Import ASCII (.dat)	2008r1	2008r1
	Import Binary (.op2)	2008r1	2008r1
	Export ASCII (.dat)	2008r1	2008r1
	Post-processing of Results	2008r1	2008r1
<b>Abaqus</b>	Import ASCII (.inp)	6.8-1	6.9-1
	Import Binary	N/A	N/A
	Export ASCII (.inp)	6.8-1	6.9-EF1
	Post-processing of Results (.fil file)	6.8-EF2	6.9-EF1
	Post-processing of Results (.odb file)	6.8-EF2	6.8-EF2
<b>ANSYS</b>	Import ASCII (PREP7, CDB)	12	12
	Import Binary (.rst, .rth)	12	12
	Export ASCII (.inp)	12	12
	Post-processing of Results	12.1	12.1
<b>LS-DYNA</b>	Import ASCII	N/A	N/A
	Import Binary	N/A	N/A
	Export ASCII (.k)	971R3.2.1	971R3.2.1
	Post-processing of Results	N/A	N/A

**NX 6 releases**

<b>Solver</b>	<b>File Type</b>	<b>NX 6</b>	<b>NX 6.0.1</b>	<b>NX 6.0.2</b>	<b>NX 6.0.3</b>	<b>NX 6.0.4</b>	<b>NX 6.0.5</b>
<b>NX Nastran</b>	Import ASCII (.dat)	6.0	6.1	6.1	6.1	6.1	7.0
	Import Binary (.op2)	6.0	6.1	6.1	6.1	6.1	7.0
	Export ASCII (.dat)	6.0	6.1	6.1	6.1	6.1	7.0
	Post-processing of Results	6.0	6.0	6.1	6.1	7.0	7.0

Solver	File Type	NX 6	NX 6.0.1	NX 6.0.2	NX 6.0.3	NX 6.0.4	NX 6.0.5
<b>MSC Nastran</b>	Import ASCII (.dat)	2007r1	2008r1	2008r1	2008r1	2008r1	2008r1
	Import Binary (.op2)	2007r1	2008r1	2008r1	2008r1	2008r1	2008r1
	Export ASCII (.dat)	2007r1	2008r1	2008r1	2008r1	2008r1	2008r1
	Post-processing of Results	2007r1	2008r1	2008r1	2008r1	2008r1	2008r1
<b>Abaqus</b>	Import ASCII (.inp)	6.7-1	6.8-1	6.8-1	6.8-1	6.8-1	6.8-1
	Import Binary	N/A	N/A	N/A	N/A	N/A	N/A
	Export ASCII (.inp)	6.7-1	6.8-1	6.8-1	6.8-1	6.8-1	6.8-1
	Post-processing of Results (.fil)	6.7-5	6.8-1	6.8-3	6.8-EF2	6.8-EF2	6.8-EF2
	Post-processing of Results (.odb)	N/A	N/A	N/A	6.8-EF	6.8-EF2	6.9-EF2
<b>ANSYS</b>	Import ASCII (PREP7, CDB)	11	11 SP1	11 SP1	11 SP1	12.0	12.0
	Import Binary (.rst, .rth)	11	11 SP1	11 SP1	11 SP1	12.0	12.0
	Export ASCII (.inp)	11	11 SP1	11 SP1	11 SP1	12.0	12.0
	Post-processing of Results	11 SP1	11 SP1	11 SP1	11 SP1	12.0	12.1
<b>LS-DYNA</b>	Import ASCII	N/A	N/A	N/A	N/A	N/A	N/A
	Import Binary	N/A	N/A	N/A	N/A	N/A	N/A
	Export ASCII (.k)	971R2	971R2	971R3.2	971R3.2	971R3.2	971R3.2.1
	Post-processing of Results	N/A	N/A	N/A	N/A	N/A	N/A

**NX 5 releases**

<b>Solver</b>	<b>File Type</b>	<b>NX 5</b>	<b>NX 5.0.1</b>	<b>NX 5.0.2</b>	<b>NX 5.0.3</b>	<b>NX 5.0.4</b>	<b>NX 5.0.5</b>	<b>NX 5.0.6</b>
<b>NX Nastran</b>	Import ASCII (.dat)	5.0	5.1	5.1	5.1	5.1	5.1	5.1
	Import Binary (.op2)	5.0	5.1	5.1	5.1	5.1	5.1	5.1
	Export ASCII (.dat)	5.0	5.1	5.1	5.1	5.1	5.1	5.1
	Post-processing of Results	5.0	5.0	5.1	5.1	5.1	5.1	6.0
<b>MSC Nastran</b>	Import ASCII (.dat)	2005	2005	2007	2007	2007	2007	2007r1
	Import Binary (.op2)	2005	2005	2007	2007	2007	2007	2007r1
	Export ASCII (.dat)	2005	2005	2007	2007	2007	2007	2007r1
	Post-processing of Results	2005	2005	2007	2007	2007	2007	2008r1
<b>Abaqus</b>	Import ASCII (.inp)	6.6	6.6	6.7-1	6.7-1	6.7-1	6.7-1	6.7-1
	Import Binary	N/A	N/A	N/A	N/A	N/A	N/A	N/A
	Export ASCII (.inp)	6.6	6.6	6.7-1	6.7-1	6.7-1	6.7-1	6.7-1
	Post-processing of Results	6.6	6.6	6.7-1	6.7-1	6.7-1	6.7-1	6.8-1
<b>ANSYS</b>	Import ASCII (PREP7, CDB)	10	10	11	11	11	11	11
	Import Binary (.rst, .rth)	10	10	11	11	11	11	11
	Export ASCII (.inp)	10	10	11	11	11	11	11
	Post-processing of Results	10	11	11	11	11	11	11 SP1



## NX 4 releases

Solver	File Type	NX 4	NX 4.0.1	NX 4.0.2	NX 4.0.3	NX 4.0.4
NX Nastran	Import ASCII (.dat)	4.0	4.1	4.1	5.0	5.0
	Import Binary (.op2)	4.0	4.1	4.1	4.1	4.1
	Export ASCII (.dat)	4.0	4.1	4.1	5.0	5.0
	Post-processing of Results	4.0	4.1	4.1	5.0	5.0
MSC Nastran	Import ASCII (.dat)	2005	2005	2005	2005	2005
	Import Binary (.op2)	2005	2005	2005	2005	2005
	Export ASCII (.dat)	2005	2005	2005	2005	2005
	Post-processing of Results	2005	2005	2005	2005	2005
Abaqus	Import ASCII (.inp)	6.5-1	6.5-1	6.5-1	6.6	6.6
	Import Binary	N/A	N/A	N/A	N/A	N/A
	Export ASCII (.inp)	6.5-1	6.5-1	6.5-1	6.6	6.6
	Post-processing of Results	6.5-1	6.5-1	6.5-1	6.6	6.6-3
ANSYS	Import ASCII (PREP7, CDB)	8	9	9	10	10
	Import Binary (.rst, .rth)	8	9	9	10	10
	Export ASCII (.inp)	8	9	9	10	10
	Post-processing of Results	9	9	9	10	10

## Define total thermal coupling conductance per element

### What is it?

You can now define the conductance of the thermal coupling at the element level. When you create a **Thermal Coupling** simulation object, you can define the total conductance value per element.

The **Per Element** check box is available only if you select **Uniform** from the **Method** list in the **Distribution** group.

You can only select the **Per Element** check box if you select **Total Conductance** from the **Type** list. If you select any other option from the **Type** list and then try to select the check box, a message appears.


**Why should I use it?**

You can import UNV files that have thermal coupling boundary conditions with conductance per element defined.

**Supported solvers and analysis types**

Solver	Analysis Type	Solution Type
NX Electronic System Cooling	Coupled Thermal-Flow	Electronic System Cooling
NX Space Systems Thermal	Thermal	Advanced Thermal/Flow with ESC NX Space Systems Thermal
NX Thermal and Flow	Thermal	Thermal
	Axisymmetric Thermal	Advanced Thermal Axisymmetric Thermal
	Coupled Thermal-Flow	Advanced Axisymmetric Thermal Thermal-Flow
		Advanced Thermal-Flow

**Where do I find it?**

Application	Advanced Simulation
Simulation Navigator	Right-click the <b>Simulation Objects</b> node ® <b>New Simulation Object</b> ® <b>Thermal Coupling</b>
Toolbar	<b>Advanced Simulation</b> ® <b>Thermal Coupling</b> 
Location in dialog box	<b>Distribution</b> group ® <b>Per Element</b> check box

**Chapter**

# *4 Customer Defaults*

**Assemblies**

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## Interface

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**ASSEMBLY  
CONSTRAINTS  
POSITIONING**

This option specifies which positioning commands are available to be used.

Version Modified: NX6.0.0

Scope: Session

Type: Integer

Default Name: Assemblies\_PositioningInteraction

Valid Options: [Mating Conditions (0), Assembly Constraints (1)]

**INTERPART COPY**

This option specifies whether or not associative interpart modeling is enabled within the Assembly Constraints dialog.

Version Modified: NX6.0.1

Scope: Session

Type: Logical

Default Name: Assemblies\_PositioningInterpartCopy

Valid Options: [Off (false), On (true)]

**CONSTRAINT  
CONVERSION  
VALIDATION  
USE CUSTOM LIMITS**

This option determines whether custom limits are used for checking component movement during constraint conversion. If this option is not set, the system bases the limits on the Positioning Linear Tolerance default.

Version Modified: NX6.0.1

Scope: Session

Type: Logical

Default Name: Assemblies\_UseCustomMateconvTolerances

Valid Options: [Yes (true), No (false)]

**MAXIMUM  
ACCEPTABLE  
COMPONENT  
MOVEMENT**

During constraint conversion, if a component would move by more than this amount a warning will be issued and the constraint will be suppressed.

Version Modified: NX6.0.1

Scope: Session  
 Type: Real  
 Default Name: Assemblies\_MateconvHardTolerance\_MU  
 Applies To: Metric  
 Units: mm  
 Valid Range: [  $\geq 1.0e-8$   $\leq 1.0e+2$  ]

**MAXIMUM  
ACCEPTABLE  
COMPONENT  
MOVEMENT**

During constraint conversion, if a component would move by more than this amount a warning will be issued and the constraint will be suppressed.

Version Modified: NX6.0.1  
 Scope: Session  
 Type: Real  
 Default Name: Assemblies\_MateconvHardTolerance\_EU  
 Applies To: English  
 Units: in  
 Valid Range: [  $\geq 1.0e-9$   $\leq 1.0e+1$  ]

**REPORT COMPONENT  
MOVEMENTS  
GREATER THAN**

During constraint conversion, if a component would move by more than this amount an informational message will be issued.

Version Modified: NX6.0.1  
 Scope: Session  
 Type: Real  
 Default Name: Assemblies\_MateconvSoftTolerance\_MU  
 Applies To: Metric  
 Units: mm  
 Valid Range: [  $\geq 1.0e-8$   $\leq 1.0e+2$  ]

**REPORT COMPONENT  
MOVEMENTS  
GREATER THAN**

During constraint conversion, if a component would move by more than this amount an informational message will be issued.

Version Modified: NX6.0.1  
Scope: Session  
Type: Real  
Default Name: Assemblies\_MateconvSoftTolerance\_EU  
Applies To: English  
Units: in  
Valid Range: [  $\geq 1.0e-9$   $\leq 1.0e+1$  ]

## Drafting

## Drawing

---

DRAWING WORK  
FLOW  
CREATE DRAFTING  
COMPONENT FOR  
VIEW OF MASTER  
MODEL PART

This option determines whether or not drafting components will be created when creating a view of the master model part.

Version Modified: NX6.0.2

Scope: Session

Type: Logical

Default Name: Drafting\_CreateComponentOfMasterModelPart

Valid Options: [Yes (true), No (false)]

---

## Annotation

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**FILL/HATCH  
RESTRICT  
CROSSHATCH TO  
+/- 45 DEGREES**

This option determines whether assembly crosshatch angles should be restricted to + or - 45 degrees.

Version Modified: NX6.0.1

Scope: Part

Type: Integer

Default Name: Axs\_restrictCrosshatchAngle

Valid Options: [On (1), Off (2)]

**WELD  
USE GB WELD  
STANDARD**

Applying this default will apply the GB standard to welds created in the part. The toggle will override any other defined weld standard. This default is obsolete in NX 7.0.0 and later releases

Version Modified: NX6.0.2

Scope: Part

Type: Logical

Default Name: Drafting\_UseGBWeldStandard

Valid Options: [No (false), Yes (true)]

**FONT**

This option specifies the line font for the weld symbol.

Version Modified: NX6.0.2

Scope: Part

Type: Integer

Default Name: Drafting\_weldSymbolFont

Valid Options: [Solid (1), Dashed (2), Phantom (3), Centerline (4), Dotted (5), Long Dashed (6), Dotted Dashed (7)]

**WIDTH**

This option specifies the line width for the weld symbol.

Version Modified: NX6.0.2

Scope: Part

Type: Integer



Default Name: Drafting\_weldSymbolWidth

Valid Options: [Thin (3), Normal (1), Thick (2)]

**COLOR**

This option specifies the color of the weld symbol.

Version Modified: NX6.0.2

Scope: Part

Type: Integer

Default Name: Drafting\_weldSymbolColor

---

## Hole Table

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### REPORT DEPTH FROM TOP AND BOTTOM FACE

This option determines whether or not to report depth from top and bottom face.

Version Modified: NX6.0.1

Scope: Session

Type: Logical

Default Name: DR\_HOLEREPORT\_FROM\_TOP\_FACE

Valid Options: [No (false), Yes (true)]

### COUNTERSINK ANGLE

This option specifies the countersink angle.

Version Modified: NX6.0.1

Scope: Session

Type: Real

Default Name: DR\_HOLEREPORT\_COUNTERSINK\_ANGLE\_MU

Applies To: Metric

Units: deg

Valid Range: [ >=0.0 ]

### COUNTERSINK ANGLE

This option specifies the countersink angle.

Version Modified: NX6.0.1

Scope: Session

Type: Real

Default Name: DR\_HOLEREPORT\_COUNTERSINK\_ANGLE\_EU

Applies To: English

Units: deg

Valid Range: [ >=0.0 ]

### PARTIAL HOLE ANGLE

This option specifies the Partial Hole angle only larger than which Partial Holes can be reported.

Version Modified: NX6.0.3

Scope: Session  
Type: Real  
Default Name: DR\_HOLEREPORT\_PARTIAL\_ANGLE  
Units: deg  
Valid Range: [ >=0.0 ]

**DISPLAY HOLE LABEL  
WITH LEADER**

This option determines whether or not to display hole label with leader.

Version Modified: NX6.0.2  
Scope: Session  
Type: Logical  
Default Name: DR\_HOLEREPORT\_LABEL\_WITH\_LEADER  
Valid Options: [Off (false), On (true)]

---

## Miscellaneous

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### GRID

This option determines which grid should be used in Drafting.

Version Modified: NX6.0.1

Scope: Session

Type: Integer

Default Name: UG\_gridInDrafting

Valid Options: [Use Drafting Grid (0), Use Sketch Grid (1)]

### SHADED VIEW FACET TOLERANCE

This option specifies the facet tolerance used for CGM creation of shaded drawing views.

Version Modified: NX6.0.5

Scope: Session

Type: Real

Default Name: Drafting\_shadedViewFacetTolerance

Valid Range: [ >=0.01 <=2.00 ]

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## Section Line

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### SECTION LINE LABEL LOCATION

This option specifies the label location for ISO128 section lines.

Version Modified: NX6.0.2

Scope: Part

Type: Integer

Default Name: Axs\_labelLocation

Valid Options: [On Arrow (0), On End (1)]

### LINE LENGTH

This option specifies the length of the end of the ESKD section line.

Version Modified: NX6.0.2

Scope: Part

Type: Real

Default Name: Axs\_lineLength\_MU

Applies To: Metric

Units: mm

Valid Range: [ >0.0 ]

### LINE LENGTH

This option specifies the length of the end of the ESKD section line.

Version Modified: NX6.0.2

Scope: Part

Type: Real

Default Name: Axs\_lineLength\_EU

Applies To: English

Units: in

Valid Range: [ >0.0 ]

---

## Detail View

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### BOUNDARY LINE COLOR

This option specifies the color of detail view boundary line.

Version Modified: NX6.0.2

Scope: Part

Type: Integer

Default Name: Drafting\_detailViewBoundaryLineColor

### BOUNDARY LINE FONT

This option specifies the detail view boundary line font.

Version Modified: NX6.0.2

Scope: Part

Type: Integer

Default Name: Drafting\_detailViewBoundaryLineFont

Valid Options: [Solid (1), Dashed (2), Phantom (3), Centerline (4), Dotted (5), Long Dashed (6), Dotted Dashed (7)]

### BOUNDARY LINE WIDTH

This option specifies the detail view boundary line width.

Version Modified: NX6.0.2

Scope: Part

Type: Integer

Default Name: Drafting\_detailViewBoundaryLineWidth

Valid Options: [Thin (2), Normal (0), Thick (1)]

---

## Flat Pattern Curves

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### TOOL MARKER ENABLED

This option specifies whether Tool Marker objects are enabled, or should be created, on flat pattern drawing views.

Version Modified: NX6.0.3

Scope: Part

Type: Logical

Default Name: NXSM\_toolMarker\_FlatPatternView\_Enabled

Valid Options: [Yes (true), No (false)]

### COLOR

This option specifies the color of Tool Marker objects on flat pattern drawing views.

Version Modified: NX6.0.3

Scope: Part

Type: Integer

Default Name: NXSM\_toolMarker\_FlatPatternView\_Color

Valid Options: [Original]

### FONT

This option specifies the font for Tool Marker objects on flat pattern drawing views.

Version Modified: NX6.0.3

Scope: Part

Type: Integer

Default Name: NXSM\_toolMarker\_FlatPatternView\_Font

Valid Options: [Original (0), Solid (1), Dashed (2), Phantom (3), Centerline (4), Dotted (5), Long Dashed (6), Dotted Dashed (7)]

### WIDTH

This option specifies the width for Tool Marker objects on flat pattern drawing views.

Version Modified: NX6.0.3

Scope: Part

Type: Integer

Default Name: NXSM\_toolMarker\_FlatPatternView\_Width

Valid Options: [Original (0), Thin (3), Normal (1), Thick (2)]



---

## Flat Pattern Annotations

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### CUSTOM CALLOUT 7 AVAILABLE

This option determines whether or not Custom Callout 7 will be available on the Preferences and Style dialog boxes for flat pattern drawings.

Version Modified: NX6.0.3

Scope: Session

Type: Logical

Default Name: NXCustomCallout\_FlatPatternView\_7\_Available

Valid Options: [Yes (true), No (false)]

### ENABLED

This option determines whether or not Custom Callout 7 will be created on flat pattern drawings.

Version Modified: NX6.0.3

Scope: Session

Type: Logical

Default Name: NXCustomCallout\_FlatPatternView\_7\_Enabled

Valid Options: [Yes (true), No (false)]

### NAME

This field defines the name that Custom Callout 7 has on the Preferences and Style dialog boxes.

Version Modified: NX6.0.3

Scope: Session

Type: String

Default Name: NXCustomCallout\_FlatPatternView\_7\_Name

### OBJECT TYPES

This field defines the object types for which Custom Callout 7 is added automatically to flat pattern views.

Version Modified: NX6.0.3

Scope: Session

Type: String

Default Name: NXCustomCallout\_FlatPatternView\_7\_AutoAttachTypes

### CONTENT

This field defines the content for Custom Callout 7.

Version Modified: NX6.0.3

Scope: Session

Type: String

Default Name: NXCustomCallout\_FlatPatternView\_7\_Content

The content can consist of any combination of expressions, special symbols, and automatic callout keyword references. An automatic callout keyword reference is replaced by the value specified by an object and a keyword name. An automatic callout keyword reference is a reference between < and > characters, like expressions, except that the leading text is !KEY. Any text outside the < and > characters may be modified. Only certain fields within the keyword reference may be modified. The number between the = and the , characters should not be changed. (No harm will befall you, it simply doesn't mean anything to change it. It references the object to which the callout will be attached when it is created.) The numbers between the , and the @ characters are the length and precision values used to format a floating point value and may be changed as needed. The keyword name must be one of the names specified by NX, and extends from the @ character to the > or a space, whichever comes first. If there are any characters between the keyword name and the > character, the keyword replacement is binary-valued. If the value returned for the keyword by the object is not zero, the first string is substituted, otherwise the second string is substituted. These strings may be modified and may contain drafting symbols or user-defined symbols.

## Flexible Printed Circuit Design

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## Curves

---

### TOOL MARKER ENABLED

This option specifies whether Tool Marker Objects are enabled, or should be created, on flat patterns.

Version Modified: NX6.0.3

Scope: Session

Type: Logical

Default Name: NXFPCB\_toolMarker\_FlatPattern\_Enabled

Valid Options: [Yes (true), No (false)]

### COLOR

This option specifies the color of Tool Marker Objects on flat patterns.

Version Modified: NX6.0.3

Scope: Session

Type: Integer

Default Name: NXFPCB\_toolMarker\_FlatPattern\_Color

### FONT

This option specifies the font for Tool Marker Objects on flat patterns.

Version Modified: NX6.0.3

Scope: Session

Type: Integer

Default Name: NXFPCB\_toolMarker\_FlatPattern\_Font

Valid Options: [Solid (1), Dashed (2), Phantom (3), Centerline (4), Dotted (5), Long Dashed (6), Dotted Dashed (7)]

### WIDTH

This option specifies the width for Tool Marker Objects on flat patterns.

Version Modified: NX6.0.3

Scope: Part

Type: Integer

Default Name: NXFPCB\_toolMarker\_FlatPattern\_Width

Valid Options: [Thin (3), Normal (1), Thick (2)]

### LAYER

This option specifies the layer on which the Tool Marker Objects will be placed in flat pattern views.

Version Modified: NX6.0.3  
Scope: Part  
Type: Integer  
Default Name: NXFPCB\_toolMarker\_FlatPattern\_Layer  
Valid Range: [ >=1 <=256 ]

## Gateway

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## Shape Search

---

### OPTION TO DEFINE PART LOCATION

This option specifies the definition of part location in result attributes.

Version Modified: NX6.0.1

Scope: Part

Type: Integer

Default Name: SHAPESEARCH\_Result\_Part\_Location\_Type

Valid Options: [One Attribute for Part Location (0), Two Attributes (Item ID + Item Revision) for Part Location (1)]

### ATTRIBUTE FOR PART LOCATION

This option specifies which result attributes will be used for part location.

Version Modified: NX6.0.1

Scope: Part

Type: Integer

Default Name: SHAPESEARCH\_Result\_Part\_Location

Valid Options: [ATTRIBUTE\_00 (0), ATTRIBUTE\_01 (1),  
ATTRIBUTE\_02 (2), ATTRIBUTE\_03 (3),  
ATTRIBUTE\_04 (4), ATTRIBUTE\_05 (5),  
ATTRIBUTE\_06 (6), ATTRIBUTE\_07 (7),  
ATTRIBUTE\_08 (8), ATTRIBUTE\_09 (9),  
ATTRIBUTE\_10 (10), ATTRIBUTE\_11 (11),  
ATTRIBUTE\_12 (12), ATTRIBUTE\_13 (13),  
ATTRIBUTE\_14 (14), ATTRIBUTE\_15 (15),  
ATTRIBUTE\_16 (16), ATTRIBUTE\_17 (17),  
ATTRIBUTE\_18 (18), ATTRIBUTE\_19 (19)]

### ATTRIBUTE FOR ITEM ID

This option specifies which result attributes will be used for item ID in Teamcenter.

Version Modified: NX6.0.1

Scope: Part

Type: Integer

Default Name: SHAPESEARCH\_Result\_Item\_ID

Valid Options: [ATTRIBUTE\_00 (0), ATTRIBUTE\_01 (1),  
ATTRIBUTE\_02 (2), ATTRIBUTE\_03 (3),  
ATTRIBUTE\_04 (4), ATTRIBUTE\_05 (5),  
ATTRIBUTE\_06 (6), ATTRIBUTE\_07 (7),  
ATTRIBUTE\_08 (8), ATTRIBUTE\_09 (9),  
ATTRIBUTE\_10 (10), ATTRIBUTE\_11 (11),  
ATTRIBUTE\_12 (12), ATTRIBUTE\_13 (13),  
ATTRIBUTE\_14 (14), ATTRIBUTE\_15 (15),  
ATTRIBUTE\_16 (16), ATTRIBUTE\_17 (17),  
ATTRIBUTE\_18 (18), ATTRIBUTE\_19 (19)]

**ATTRIBUTE FOR ITEM  
REVISION**

This option specifies which result attributes will be used for item revision in Teamcenter.

Version Modified: NX6.0.1

Scope: Part

Type: Integer

Default Name: SHAPESEARCH\_Result\_Item\_Revision

Valid Options: [ATTRIBUTE\_00 (0), ATTRIBUTE\_01 (1),  
ATTRIBUTE\_02 (2), ATTRIBUTE\_03 (3),  
ATTRIBUTE\_04 (4), ATTRIBUTE\_05 (5),  
ATTRIBUTE\_06 (6), ATTRIBUTE\_07 (7),  
ATTRIBUTE\_08 (8), ATTRIBUTE\_09 (9),  
ATTRIBUTE\_10 (10), ATTRIBUTE\_11 (11),  
ATTRIBUTE\_12 (12), ATTRIBUTE\_13 (13),  
ATTRIBUTE\_14 (14), ATTRIBUTE\_15 (15),  
ATTRIBUTE\_16 (16), ATTRIBUTE\_17 (17),  
ATTRIBUTE\_18 (18), ATTRIBUTE\_19 (19)]

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## JT Files

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### IMPORT OPTIONS USE WORK LAYER

This option determines whether or not the work layer should be used for model geometry objects (solids, sheets, faceted bodies) when opening JT files in NX. If this option is turned off, the objects are placed on the layer as specified by the Model Geometry Layer option.

Version Modified: NX6.0.2

Scope: Session

Type: Logical

Default Name: Gateway\_JT\_import\_model\_geometry\_layer\_option

Valid Options: [No (false), Yes (true)]

### MODEL GEOMETRY LAYER

This option specifies what layer model geometry objects (solids, sheets, faceted bodies) will be placed on when opening JT files in NX. This will only have effect if the Use Work Layer option is toggled off.

Version Modified: NX6.0.2

Scope: Session

Type: Integer

Default Name: Gateway\_JT\_import\_model\_geometry\_layer

Valid Range: [ >=1 <=256 ]

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## Reuse Library

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**SHOW ATTRIBUTE  
SEARCH FIRST FOR  
CLASSIFICATION  
NODE**

This option determines whether or not the Reuse Library navigator will show attribute search when a classification node is selected.

Version Modified: NX6.0.2

Scope: Session

Type: Logical

Default Name: KRU\_Classification\_Attribute\_Search\_First

Valid Options: [No (false), Yes (true)]



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## Materials

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### NX MATERIAL LIBRARY ENABLE

This option specifies whether materials in the legacy NX Material Library will be shown in the list on material dialog boxes.

Version Modified: NX6.0.2

Scope: Session

Type: Integer

Default Name: UG\_NxMaterialLibraryEnable

Valid Options: [Yes (1), No (0)]

### ALLOW USER CONTROL

This option specifies whether the 'NX Material Library' check box on material dialog boxes is allowed to be changed. When this option is turned off and the above 'Enable' option is turned on, the 'NX Material Library' check box will be disabled. When this option is turned off and the above 'Enable' option is turned off, the 'NX Material Library' check box will be hidden.

Version Modified: NX6.0.2

Scope: Session

Type: Integer

Default Name: UG\_NxMaterialLibraryAllowUserControl

Valid Options: [Yes (1), No (0)]

### SITE MATML LIBRARY ENABLE

This option specifies whether the materials in the specified Site MatML Library file will be shown in the list on material dialog boxes.

Version Modified: NX6.0.2

Scope: Session

Type: Integer

Default Name: UG\_SiteMatMLLibraryEnable

Valid Options: [Yes (1), No (0)]

### LIBRARY FILE NAME

This option specifies the Site MatML library file name.

Version Modified: NX6.0.2

Scope: Session

Type: String  
Default Name: UG\_SiteMatMLLibrary\_UNX  
Applies To: Unix

**LIBRARY FILE NAME**

This option specifies the Site MatML library file name.

Version Modified: NX6.0.2  
Scope: Session  
Type: String  
Default Name: UG\_SiteMatMLLibrary\_WIN  
Applies To: Windows

**ALLOW USER CONTROL**

This option specifies whether the 'Site MatML Library' check box on material dialog boxes is allowed to be changed. When this option is turned off and the above 'Enable' option is turned on, the 'Site MatML Library' check box and the library file specification will be disabled. When this option is turned off and the above 'Enable' option is turned off, the 'Site MatML Library' check box and the library file specification will be hidden.

Version Modified: NX6.0.2  
Scope: Session  
Type: Integer  
Default Name: UG\_SiteMatMLLibraryAllowUserControl  
Valid Options: [Yes (1), No (0)]

**USER MATML LIBRARY ENABLE**

This option specifies whether the materials in the specified User MatML Library file will be shown in the list on material dialog boxes.

Version Modified: NX6.0.2  
Scope: Session  
Type: Integer  
Default Name: UG\_UserMatMLLibraryEnable  
Valid Options: [Yes (1), No (0)]

**LIBRARY FILE NAME**

This option specifies the User MatML library file name.

Version Modified: NX6.0.2  
Scope: Session

Type: String  
Default Name: UG\_UserMatMLLibrary\_UNX  
Applies To: Unix

**LIBRARY FILE NAME**

This option specifies the User MatML library file name.

Version Modified: NX6.0.2  
Scope: Session  
Type: String  
Default Name: UG\_UserMatMLLibrary\_WIN  
Applies To: Windows

**ALLOW USER CONTROL**

This option specifies whether the 'User MatML Library' check box on material dialog boxes is allowed to be changed. When this option is turned off and the above 'Enable' option is turned on, the 'User MatML Library' check box and the library file specification will be disabled. When this option is turned off and the above 'Enable' option is turned off, the 'User MatML Library' check box and the library file specification will be hidden.

Version Modified: NX6.0.2  
Scope: Session  
Type: Integer  
Default Name: UG\_UserMatMLLibraryAllowUserControl  
Valid Options: [Yes (1), No (0)]

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## View Sectioning

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**LOAD SOLIDS/SHEETS  
WHEN SAVING  
SECTION CURVES**

When this option is set, the Save Copy of Section Curves command in view sectioning and the datum plane grid dialog will cause solid/sheet bodies to be loaded into memory for any visible lightweight bodies on the section plane. This may increase the time and memory used by the operation, but will ensure fully accurate section curves.

Version Modified: NX6.0.4  
Scope: Session  
Type: Integer  
Default Name: UG\_viewSectioningLoadOnSave  
Valid Options: [Off (0), On (1)]

## Manufacturing

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## General

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### ALLOW OVERWRITING OF LOCKED TOOL PATHS

This option determines whether or not users can overwrite a tool path when the Tool Path Overwrite message is displayed. When this is off, users must explicitly unlock the tool path before performing an operation that overwrites it.

Version Modified: NX6.0.4

Scope: Session

Type: Logical

Default Name: CAM\_lock\_path\_allow\_overwrite

Valid Options: [Yes (true), No (false)]

This option determines whether or not a minimum damping ratio should be used. When this check box is turned on, a minimum damping ratio (when damping is not already defined in the event) of 0.001 percent is used. When this check box is turned off, the damping value defaults to zero.

## Modeling

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## Miscellaneous

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**ALLOW CHANGING  
TO HISTORY-FREE  
MODELING MODE**

This option determines whether or not it is allowed to change to History-Free modeling mode.

Version Modified: NX6.0.2

Scope: Session

Type: Logical

Default Name: UGII\_ALLOW\_HISTORY\_FREE

Valid Options: [No (false), Yes (true)]

## Mold Wizard

---

## Pocket

---

### POCKET COLOR SETTING

This option specifies where the pocket faces color inherits.

Version Modified: NX6.0.4

Scope: Session

Type: Integer

Default Name: MW\_POCKET\_INHERIT\_COLOR

Valid Options: [Tool Face Color (0), Tool Body Color (1), Target Body Color (2), Modeling Default (3)]

### LAYER CONTROL

This option determines whether or not the pocket creation will be controlled by 1  
If the option is on, the tool bodies which are in invisible layer will be ignored  
Otherwise, whether or not the tool bodies are in invisible layer, they will be us

Version Modified: NX6.0.5

Scope: Session

Type: Logical

Default Name: MW\_POCKET\_LAYER\_CONTROL

Valid Options: [No (false), Yes (true)]

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## Teamcenter

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### TEAMCENTER FOLDER SEARCHING

This option specifies if the folders will be searched under the home folder.

Version Modified: NX6.0.1

Scope: Session

Type: Integer

Default Name: MW\_IMAN\_FIND\_FOLDER

Valid Options: [Never (0), Home Folder (1)]

### TEAMCENTER FOLDER SEARCHING

This option specifies if the folders will be searched under the home folder.

Version Modified: NX6.0.1

Scope: Session

Type: Integer

Default Name: DE\_MW\_IMAN\_FIND\_FOLDER

Valid Options: [Never (0), Home Folder (1)]

### TEAMCENTER FOLDER SEARCHING

This option specifies if the folders will be searched under the home folder.

Version Modified: NX6.0.1

Scope: Session

Type: Integer

Default Name: MW\_EW\_IMAN\_FIND\_FOLDER

Valid Options: [Never (0), Home Folder (1)]

## Motion



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## Adams Solve Method

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### ARTICULATION SIMULATION TYPE

This option specifies the Adams simulation method to solve with for articulation analysis.

Version Modified: NX6.0.0

Scope: Session

Type: String

Default Name: CAE\_DefaultMotionArticulationAdamsSimType

Valid Options: [Quasi Static (QuasiStatic), Dynamic (Dynamic)]

---

## Point on Curve

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**CURVE  
PARAMETERIZATION  
METHOD**

This option specifies the curve spacing method for Point on Curve.

Version Modified: NX6.0.1

Scope: Part

Type: String

Default Name: CAE\_DefaultMotionCurveParameterizationMethod

Valid Options: [User-defined Spacing (UserDefinedSpacing),  
Curvature-based (CurvatureBased)]

**SPACING**

This option specifies the initial arc length.

Version Modified: NX6.0.1

Scope: Part

Type: Real

Default Name: CAE\_DefaultMotionCurveSpacing\_MU

Applies To: Metric

Units: mm

Valid Range: [ >0.0 ]

**SPACING**

This option specifies the initial arc length.

Version Modified: NX6.0.1

Scope: Part

Type: Real

Default Name: CAE\_DefaultMotionCurveSpacing\_EU

Applies To: English

Units: in

Valid Range: [ >0.0 ]

## Product Validation

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## Check Requirements

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### SOURCE TYPE FOR EXTERNAL REQUIREMENTS

This option specifies the database to be used as the source for external requirements.

Version Modified: NX6.0.2

Scope: Session

Type: Integer

Default Name: CheckRequirements\_sourceType

Valid Options: [Teamcenter (0), TcSE (1), Spreadsheet or XML File (2)]

### ALLOW FILES AS ADDITIONAL REQUIREMENTS SOURCES IN TEAMCENTER OR TCSE

This option specifies whether spreadsheets and XML files may be used as additional sources when the requirements source type is Teamcenter or TcSE.

Version Modified: NX6.0.2

Scope: Session

Type: Integer

Default Name: CheckRequirements\_allowAdditionalSources

Valid Options: [Yes (1), No (0)]

### DEFAULT REQUIREMENTS REVISION RULE

This option specifies the default revision rule for requirements from Teamcenter.

Version Modified: NX6.0.2

Scope: Session

Type: String

Default Name: CheckRequirements\_defaultTeamcenterReqRevisionRule

### SHOW REQUIREMENT CHECK MESSAGE

This option determines whether or not to show a requirement check message and if so, at what severity levels.

Version Modified: NX6.0.2

Scope: Session  
Type: Integer  
Default Name: CheckRequirements\_showMessagewhenCheckFail  
Valid Options: [No (0), Yes, but only on Fail (1), Yes, on Fail or Warning (2), Yes, on Fail, Warning, or Information (3)]

**REFRESH OUTDATED  
REQUIREMENTS  
WHEN PART IS  
OPENED**

This option specifies whether the outdated requirements will be refreshed when part is opened.

Version Modified: NX6.0.3  
Scope: Session  
Type: Integer  
Default Name: CheckRequirements\_updateReqonPartOpen  
Valid Options: [Yes (1), No (0)]

## **Progressive Die Wizard**

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## Pocket

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### POCKET COLOR SETTING

This option specifies where the pocket faces color inherits.

Version Modified: NX6.0.4

Scope: Session

Type: Integer

Default Name: PDW\_POCKET\_INHERIT\_COLOR

Valid Options: [Tool Face Color (0), Tool Body Color (1), Target Body Color (2), Modeling Default (3)]

### LAYER CONTROL

This option determines whether or not the pocket creation will be controlled by 1  
If the option is on, the tool bodies which are in invisible layer will be ignored  
Otherwise, whether or not the tool bodies are in invisible layer, they will be us

Version Modified: NX6.0.5

Scope: Session

Type: Logical

Default Name: PDW\_POCKET\_LAYER\_CONTROL

Valid Options: [No (false), Yes (true)]

---

## Teamcenter

---

### TEAMCENTER FOLDER SEARCHING

This option specifies if the folders will be searched under the home folder.

Version Modified: NX6.0.1

Scope: Session

Type: Integer

Default Name: PDW\_IMAN\_FIND\_FOLDER

Valid Options: [Never (0), Home Folder (1)]

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## Punch False Body

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The following options specify the color, font, width and layer of the punch false body.

**COLOR**

Version Modified: NX6.0.1  
 Scope: Session  
 Type: Integer  
 Default Name: PDW\_FALSE\_COLOR

**FONT**

Version Modified: NX6.0.1  
 Scope: Session  
 Type: Integer  
 Default Name: PDW\_FALSE\_LINE\_FONT  
 Valid Options: [Invisible (0), Solid (1), Dashed (2), Phantom (3), Centerline (4), Dotted (5), Long Dashed (6), Dotted Dashed (7)]

**WIDTH**

Version Modified: NX6.0.1  
 Scope: Session  
 Type: Integer  
 Default Name: PDW\_FALSE\_LINE\_WIDTH  
 Valid Options: [No Change (-1), Thin (2), Normal (0), Thick (1)]

**LAYER**

Version Modified: NX6.0.1  
 Scope: Session  
 Type: Integer  
 Default Name: PDW\_FALSE\_LAYER  
 Valid Range: [ >=1 <=256 ]

## Ship Design

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## Manufacturing XML Output

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The following options specify the values for the parameters for Ship Concept Manufacturing XML Output.

**OUTPUT  
SPREADSHEET FILE**

This field specifies the spreadsheet output file. The directory containing this file is the base directory for the XML output files.

Version Modified: NX6.0.0

Scope: Part

Type: String

Default Name: PMAN\_Ship\_Concept\_Manufacturing\_Output\_XLS\_File\_WIN

Applies To: Windows

**OUTPUT  
SPREADSHEET FILE**

This field specifies the spreadsheet output file. The directory containing this file is the base directory for the XML output files.

Version Modified: NX6.0.0

Scope: Part

Type: String

Default Name: PMAN\_Ship\_Concept\_Manufacturing\_Output\_XLS\_File\_UNX

Applies To: Unix

## Sheet Metal



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## Curves

---

### TOOL MARKER ENABLED

This option specifies whether Tool Marker Objects are enabled, or should be created, on flat patterns.

Version Modified: NX6.0.3

Scope: Session

Type: Logical

Default Name: NXSM\_toolMarker\_FlatPattern\_Enabled

Valid Options: [Yes (true), No (false)]

### COLOR

This option specifies the color of Tool Marker Objects on flat patterns.

Version Modified: NX6.0.3

Scope: Session

Type: Integer

Default Name: NXSM\_toolMarker\_FlatPattern\_Color

### FONT

This option specifies the font for Tool Marker Objects on flat patterns.

Version Modified: NX6.0.3

Scope: Session

Type: Integer

Default Name: NXSM\_toolMarker\_FlatPattern\_Font

Valid Options: [Solid (1), Dashed (2), Phantom (3), Centerline (4), Dotted (5), Long Dashed (6), Dotted Dashed (7)]

### WIDTH

This option specifies the width for Tool Marker Objects on flat patterns.

Version Modified: NX6.0.3

Scope: Part

Type: Integer

Default Name: NXSM\_toolMarker\_FlatPattern\_Width

Valid Options: [Thin (3), Normal (1), Thick (2)]

### LAYER

This option specifies the layer on which the Tool Marker Objects will be placed in flat pattern views.

Version Modified: NX6.0.3  
Scope: Part  
Type: Integer  
Default Name: NXSM\_toolMarker\_FlatPattern\_Layer  
Valid Range: [ >=1 <=256 ]

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## Annotations

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### CUSTOM CALLOUT 7 AVAILABLE

This option determines whether or not Custom Callout 7 will be available on the Preferences and Style dialog boxes for flat patterns.

Version Modified: NX6.0.3

Scope: Session

Type: Logical

Default Name: NXCustomCallout\_FlatPattern\_7\_Available

Valid Options: [Yes (true), No (false)]

### ENABLED

This option determines whether or not Custom Callout 7 will be created on flat patterns.

Version Modified: NX6.0.3

Scope: Session

Type: Logical

Default Name: NXCustomCallout\_FlatPattern\_7\_Enabled

Valid Options: [Yes (true), No (false)]

### NAME

This field defines the name that Custom Callout 7 has on the Preferences and Style dialog boxes.

Version Modified: NX6.0.3

Scope: Session

Type: String

Default Name: NXCustomCallout\_FlatPattern\_7\_Name

### OBJECT TYPES

This field defines the object types for which Custom Callout 7 is valid.

Version Modified: NX6.0.3

Scope: Session

Type: String

Default Name: NXCustomCallout\_FlatPattern\_7\_AutoAttachTypes

### CONTENT

This field defines the content for Custom Callout 7.

Version Modified: NX6.0.3

Scope: Session

Type: String

Default Name: NXCustomCallout\_FlatPattern\_7\_Content

The content can consist of any combination of expressions, special symbols, and automatic callout keyword references. An automatic callout keyword reference is replaced by the value specified by an object and a keyword name. An automatic callout keyword reference is a reference between < and > characters, like expressions, except that the leading text is !KEY. Any text outside the < and > characters may be modified. Only certain fields within the keyword reference may be modified. The number between the = and the , characters should not be changed. (No harm will befall you, it simply doesn't mean anything to change it. It references the object to which the callout will be attached when it is created.) The numbers between the , and the @ characters are the length and precision values used to format a floating point value and may be changed as needed. The keyword name must be one of the names specified by NX, and extends from the @ character to the > or a space, whichever comes first. If there are any characters between the keyword name and the > character, the keyword replacement is binary-valued. If the value returned for the keyword by the object is not zero, the first string is substituted, otherwise the second string is substituted. These strings may be modified and may contain drafting symbols or user-defined symbols.

## Simulation

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## General

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### MERGE PHYSICAL PROPERTIES ON IMPORT

This option determines whether or not identical physical properties will be merged during import. This option results in a better performance on import at the expense of a more detailed breakdown of the model.

Version Modified: NX6.0.1

Scope: Part

Type: Logical

Default Name: CAE\_DefaultMergePptImport

Valid Options: [On (true), Off (false)]

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## Meshing

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### ATTEMPT QUAD ONLY (SWEPT MESH DIALOG)

For the surface mesh on the source face of a 3D swept mesh, this option controls whether the mesh will contain only quadrilateral elements. Off-Allow Triangles: Allows triangular elements in the mesh. On-Zero Triangles: Prevents the creation of triangular elements in the mesh. On-Single Triangle: Creates at most a single triangular element on the source face.

Version Modified: NX6.0.1

Scope: Part

Type: Integer

Default Name: CAE\_DefaultQuadOnlyHexMesh

Valid Options: [Off - Allow Triangles (0), On - Zero Triangles (1), On - Single Triangle (2)]

### ATTEMPT QUAD ONLY (2D MESH DIALOG)

For a 2D free mesh, this option controls whether the mesh will contain only quadrilateral elements. Off-Allow Triangles: Allows triangular elements in the mesh. On-Zero Triangles: Prevents the creation of triangular elements in the mesh. On-Single Triangle: Creates at most a single triangular element per face.

Version Modified: NX6.0.1

Scope: Part

Type: Integer

Default Name: CAE\_DefaultQuadOnlyFreeMesh

Valid Options: [Off - Allow Triangles (0), On - Zero Triangles (1), On - Single Triangle (2)]

### QUAD ONLY FOR 3 SIDED FACES (MAPPED MESH DIALOG)

This option controls whether an all quadrilateral mesh is generated for faces that have only three sides.

Version Modified: NX6.0.1

Scope: Part

Type: Logical

Default Name: CAE\_DefaultQuadOnlyMappedMesh

Valid Options: [On (true), Off (false)]

STORE FREE MESH  
IF MAPPED MESH  
FAILS (MAPPED MESH  
DIALOG)

In the event a mapped mesh cannot be generated on a given face, this option controls whether the software stores the free mesh that was previewed on the face.

Version Modified: NX6.0.1

Scope: Part

Type: Logical

Default Name: CAE\_DefaultKeepFreeMeshesMappedMesh

Valid Options: [On (true), Off (false)]

---

## **NASTRAN**

---

### **STRUCTURAL OUTPUT REQUESTS GASKET RESULTS**

This option determines whether or not the GKRESULTS request will be enabled when creating a Structural Output Request modeling object in Advanced Simulation.

Version Modified: NX6.0.1

Scope: Part

Type: Logical

Default Name: CAE\_DefaultGasketResultRequest

Valid Options: [Yes (true), No (false)]

### **NONLINEAR STRESS**

This option determines whether or not the NLSTRESS request will be enabled when creating a Structural Output Request modeling object in Advanced Simulation.

Version Modified: NX6.0.1

Scope: Part

Type: Logical

Default Name: CAE\_DefaultNonlinearStressRequest

Valid Options: [Yes (true), No (false)]

### **SHELL THICKNESS**

This option determines whether or not the SHELLTHK request will be enabled when creating a Structural Output Request modeling object in Advanced Simulation.

Version Modified: NX6.0.2

Scope: Part

Type: Logical

Default Name: CAE\_DefaultShellThicknessRequest

Valid Options: [Yes (true), No (false)]



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## ABAQUS

---

### RESULTS FILE ABAQUS FILE EXTENSION

This option specifies the file extension of the ABAQUS results file that NX should load. This also specifies whether NX will load ABAQUS odb related shared objects, which might slow down the file reading.

Version Modified: NX6.0.2

Scope: Session

Type: String

Default Name: CAE\_Vkitype\_WIN

Applies To: Windows

Valid Options: [FIL (fil), ODB (odb)]

### Fields Extras

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## Basic Display

---

### COLOR

This option specifies the basic color of fields.

Version Modified: NX6.0.1

Scope: Part

Type: Integer

Default Name: CAE\_DefaultFieldColor

### LINE FONT

This option specifies the line font for fields.

Version Modified: NX6.0.1

Scope: Part

Type: Integer

Default Name: CAE\_DefaultFieldLineFont

Valid Options: [Solid (0), Dashed (1), Phantom (2), Centerline (3), Dotted (4), Long Dashed (5), Dotted Dashed (6)]

### LINE WIDTH

This option specifies the line width for fields.

Version Modified: NX6.0.1

Scope: Part

Type: Integer

Default Name: CAE\_DefaultFieldLineWidth

Valid Options: [Thin (2), Normal (0), Thick (1)]

### RESOLUTION

This option specifies the density of the point sampling for field display.

Version Modified: NX6.0.1

Scope: Part

Type: Integer

Default Name: CAE\_DefaultFieldResolution

Valid Options: [Coarse (0), Standard (1), Fine (2), Extra Fine (3), Super Fine (4), Ultra Fine (5)]

### TRANSLUCENCY

This option determines the translucency of shaded display for fields.

Version Modified: NX6.0.1

---

Scope: Part  
Type: Integer  
Default Name: CAE\_DefaultFieldTranslucency  
Units: %  
Valid Range: [ >=0 <=100 ]

**PARTIALLY SHADED**

This option determines whether or not shaded portions of field display are drawn in Partially Shaded mode.

Version Modified: NX6.0.1  
Scope: Part  
Type: Logical  
Default Name: CAE\_DefaultFieldPartiallyShaded  
Valid Options: [On (true), Off (false)]

**FACE ANALYSIS**

This option determines whether or not shaded portions of field display are drawn in Face Analysis mode.

Version Modified: NX6.0.1  
Scope: Part  
Type: Logical  
Default Name: CAE\_DefaultFieldFaceAnalysis  
Valid Options: [On (true), Off (false)]

---

## Independent Domain Display

---

### DISPLAY TYPE

This option specifies the display type for the independent domain points for field display.

Version Modified: NX6.0.1

Scope: Part

Type: Integer

Default Name: CAE\_DefaultFieldIndepDispType

Valid Options: [Normal (0), Point (1), Plus Sign (2), Asterisk (3), Circle (4), Pound Sign (5), Cross (6), Square (7), Triangle (8), Diamond (9), Centerline (10), Hide (11)]

### AXES

This option determines if the axes for the independent domain are displayed.

Version Modified: NX6.0.1

Scope: Part

Type: Logical

Default Name: CAE\_DefaultFieldShowAxes

Valid Options: [On (true), Off (false)]

### AXES COLOR

This option specifies the color of independent domain axes if displayed.

Version Modified: NX6.0.1

Scope: Part

Type: Integer

Default Name: CAE\_DefaultFieldAxesColor

### LABELS

This option determines if the labels for the field are displayed; if the axes are displayed and this option is turned on, the axes labels will be also displayed.

Version Modified: NX6.0.1

Scope: Part

Type: Logical

Default Name: CAE\_DefaultFieldShowLabels

Valid Options: [On (true), Off (false)]

### LABEL COLOR

This option specifies the color of field labels.

Version Modified: NX6.0.1  
Scope: Part  
Type: Integer  
Default Name: CAE\_DefaultFieldLabelColor

---

## Dependent Domain Display

---

### DISPLAY TYPE

This option specifies the display type for the dependent domain.

Version Modified: NX6.0.1

Scope: Part

Type: Integer

Default Name: CAE\_DefaultFieldDepDispType

Valid Options: [Hide (2), Symbol (0), Surface (1), Surface With Edges (3)]

### SYMBOL SIZE

This option specifies the symbol size when in Symbol mode.

Version Modified: NX6.0.1

Scope: Part

Type: Real

Default Name: CAE\_DefaultFieldSymbolSize

Valid Range: [ >=0.0 <=25.0 ]

### SURFACE OFFSET

This option specifies the offset of the dependent surface from the independent domain when in Surface mode.

Version Modified: NX6.0.1

Scope: Part

Type: Real

Default Name: CAE\_DefaultFieldSurfaceOffset

Valid Range: [ >=0.0 <=50.0 ]

### COLOR VALUE

This option specifies the color option for the dependent domain.

Version Modified: NX6.0.1

Scope: Part

Type: Integer

Default Name: CAE\_DefaultFieldDepColorValue

Valid Options: [Spectrum (2), Inherit (0), Specified (1)]

### SPECIFIED COLOR

This option specifies the dependent domain color when Color Value is Specified.

Version Modified: NX6.0.1  
Scope: Part  
Type: Integer  
Default Name: CAE\_DefaultFieldDepColor

**LABELS**

This option specifies which dependent values to display.

Version Modified: NX6.0.1  
Scope: Part  
Type: Integer  
Default Name: CAE\_DefaultFieldValueLabels  
Valid Options: [None (0), Minimum / Maximum (1), Maximum (2), Minimum (3), All (4)]

**UNDEFINED VALUES**

This option specifies how to display undefined values.

Version Modified: NX6.0.1  
Scope: Part  
Type: Integer  
Default Name: CAE\_DefaultFieldUndefDispType  
Valid Options: [Hide (0), Point (1), Plus Sign (2), Asterisk (3), Circle (4), Pound Sign (5), Cross (6), Square (7), Triangle (8), Diamond (9), Centerline (10)]

**UNDEFINED COLOR**

This option specifies the color of undefined values if they are displayed.

Version Modified: NX6.0.1  
Scope: Part  
Type: Integer  
Default Name: CAE\_DefaultFieldUndefinedColor

---

## Table Display

---

### INDEPENDENT POINT DISPLAY

This option specifies how to display the independent domain points for a table.

Version Modified: NX6.0.1

Scope: Part

Type: Integer

Default Name: CAE\_DefaultFieldTableIndepDispType

Valid Options: [Hide (0), Point (1), Plus Sign (2), Asterisk (3), Circle (4), Pound Sign (5), Cross (6), Square (7), Triangle (8), Diamond (9), Centerline (10)]

### INDEPENDENT DOMAIN COLOR

This option specifies the color of independent domain points for a table when they are displayed.

Version Modified: NX6.0.1

Scope: Part

Type: Integer

Default Name: CAE\_DefaultFieldTableIndepColor

## Routing



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## Extras

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### HEAL PATH CURVE CREATE SMART CONTROL POINTS

This option determines whether or not smart control points are created when creating a Heal Path Spline segment.

Version Modified: NX5.0.6

Scope: Session

Type: Logical

Default Name: RTE\_PathHealPathSplineSmartControlPointType

Valid Options: [Yes (true), No (false)]

### USE INTERPART EXTRACTS

This option determines whether or not to use interpart extracts when creating a Heal Path Spline segment.

Version Modified: NX5.0.6

Scope: Session

Type: Logical

Default Name: RTE\_PathHealPathSplineInterpartExtractsType

Valid Options: [Yes (true), No (false)]

### OVERSTOCK MINIMUM OVERSTOCK WIDTH (ROUTING ELECTRICAL)

This option specifies the minimum overstock length for overstock that does not have a specified width, in Routing Electrical application

Version Modified: NX6.0.2

Scope: Session

Type: Real

Default Name: RTE\_ElectricalMinimumOverstockWidth\_MU

Applies To: Metric

Units: mm

Valid Range: [ >0.0 ]

**MINIMUM OVERSTOCK  
WIDTH (ROUTING  
ELECTRICAL)**

This option specifies the minimum overstock length for overstock that does not have a specified width, in Routing Electrical application

Version Modified: NX6.0.2

Scope: Session

Type: Real

Default Name: RTE\_ElectricalMinimumOverstockWidth\_EU

Applies To: English

Units: in

Valid Range: [ >0.0 ]

**MINIMUM OVERSTOCK  
WIDTH (ROUTING  
MECHANICAL)**

This option specifies the minimum overstock length for overstock that does not have a specified width, in Routing Mechanical application

Version Modified: NX6.0.2

Scope: Session

Type: Real

Default Name: RTE\_MechanicalMinimumOverstockWidth\_MU

Applies To: Metric

Units: mm

Valid Range: [ >0.0 ]

**MINIMUM OVERSTOCK  
WIDTH (ROUTING  
MECHANICAL)**

This option specifies the minimum overstock length for overstock that does not have a specified width, in Routing Mechanical application

Version Modified: NX6.0.2

Scope: Session

Type: Real

Default Name: RTE\_MechanicalMinimumOverstockWidth\_EU

Applies To: English

Units: in

Valid Range: [ >0.0 ]

**ALLOW OVERSTOCK  
DEFINING CONTROL  
POINTS TO BE  
SIMPLIFIED**

This option specifies whether or not an overstock's defining control point is simplified away when using Simplify Path.

Version Modified: NX6.0.2

Scope: Session

Type: Logical

Default Name: RTE\_OverstockSimplification

Valid Options: [No (false), Yes (true)]

When simplifying a path with overstock on the path, overstock defining control points may be simplified away. When this default is on, the overstock is deleted when the defining control point is removed. When this default is off, the defining control point is not simplified away and the overstock remains on the path.

**EDIT BEND ANGLE  
OPTIONS  
EDIT BEND ANGLE**

This option specifies which side of the bend arc the angle between the segments is measured in the Edit Bend Angle command.

Version Modified: NX6.0.5

Scope: Session

Type: Integer

Default Name: RTE\_EditBendAngleMeasureInteriorAngle

Valid Options: [Measure Interior Angle (1), Measure Exterior Angle (2)]

## Teamcenter Integration

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## Issue Navigator - Teamcenter Community

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**TEAMCENTER  
COMMUNITY SITE URL  
OR ISSUE LIST URL**

This option specifies the URL of the Teamcenter Community Site or Issue List.

Version Modified: NX6.0.3

Scope: Session

Type: String

Default Name: Issue\_TccUrl

**USER DEFINED  
MANDATORY  
COLUMNS**

Issue Navigator requires some mandatory columns defined in Teamcenter Community Issue List: ID Issue ID Title Comment Assigned To Status Due Date Lock State Priority Image NX Reserved 1 NX Reserved 2 NX Reserved 3 NX Reserved 4 NX Reserved 5 NX Reserved 6 User can add more columns if those columns are mandatory in user's site. This option is to let user define additional mandatory columns which must be defined in Teamcenter Community Issue List.

Version Modified: NX6.0.3

Scope: Session

Type: String

Default Name: Issue\_TccMandatoryColumns

Check the Teamcenter Community Issue List settings for column names. If the column name specified in this option is not defined in Teamcenter Community Issue List, Issue Navigator will display the missing column information dialog and stop the connection.

**OPTIONAL COLUMNS**

This option specifies the optional columns which can be defined in Teamcenter Community Issue List.

Version Modified: NX6.0.3

Scope: Session

Type: String

Default Name: Issue\_TccOptionalColumns

Check the Teamcenter Community Issue List settings for column names. If the column name specified in this option is not defined in Teamcenter

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Community Issue List, Issue Navigator will ignore this column and continue the connection.

**FILTER CONDITONS**

This option specifies the filter conditions to retrieve only the satisfied issues from Teamcenter Community Issue List. Format of the filter conditions is: column name=column value; or column name!=column value. For example: Status=Active.

Version Modified: NX6.0.3  
Scope: Session  
Type: String  
Default Name: Issue\_TccFilterConditions

**EMAIL NOTIFICATION**

This option specifies whether there is email notification when the issue is created or updated.

Version Modified: NX6.0.3  
Scope: Session  
Type: Logical  
Default Name: Issue\_TccEmailNotification  
Valid Options: [On (true), off (false)]

**NATIVE NX  
DOWNLOAD  
ATTACHMENTS WHEN  
RETRIEVING ISSUES**

This option specifies whether the attachments will be downloaded to local disk when the issues are retrieved from Teamcenter Community Issue List.

Version Modified: NX6.0.3  
Scope: Session  
Type: Logical  
Default Name: Issue\_TccDownloadAllAttachments  
Valid Options: [On (true), off (false)]

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## Intellectual Property Protection

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The following options specify which commands will be blocked and logged, and which ones will be logged only.

**FILE -> SAVE AS**

Version Modified: NX6.0.1  
Scope: Session  
Type: Integer  
Default Name: UG\_IP\_menu\_file\_saveas  
Valid Options: [Block (1), Log (0)]

**FILE -> EXPORT  
ASSEMBLY OUTSIDE  
TEAMCENTER**

Version Modified: NX6.0.1  
Scope: Session  
Type: Integer  
Default Name: UG\_IP\_menu\_file\_export\_assembly  
Valid Options: [Block (1), Log (0)]

**FILE -> PRINT**

Version Modified: NX6.0.1  
Scope: Session  
Type: Integer  
Default Name: UG\_IP\_menu\_file\_print  
Valid Options: [Block (1), Log (0)]

**FILE -> PLOT**

Version Modified: NX6.0.1  
Scope: Session  
Type: Integer  
Default Name: UG\_IP\_menu\_file\_plot  
Valid Options: [Block (1), Log (0)]

**FILE -> SEND TO  
PACKAGE FILE**

Version Modified: NX6.0.1

Scope: Session  
Type: Integer  
Default Name: UG\_IP\_menu\_file\_send  
Valid Options: [Block (1), Log (0)]

**FILE -> EXPORT ->  
PART**

Version Modified: NX6.0.1  
Scope: Session  
Type: Integer  
Default Name: UG\_IP\_menu\_export\_part  
Valid Options: [Block (1), Log (0)]

**FILE -> EXPORT ->  
PARASOLID**

Version Modified: NX6.0.1  
Scope: Session  
Type: Integer  
Default Name: UG\_IP\_menu\_export\_parasolid  
Valid Options: [Block (1), Log (0)]

**FILE -> EXPORT  
-> USER DEFINED  
FEATURE**

Version Modified: NX6.0.1  
Scope: Session  
Type: Integer  
Default Name: UG\_IP\_menu\_export\_user  
Valid Options: [Block (1), Log (0)]

**FILE -> EXPORT  
-> DRAWINGS TO  
TEAMCENTER**

Version Modified: NX6.0.1  
Scope: Session  
Type: Integer  
Default Name: UG\_IP\_menu\_export\_drawings  
Valid Options: [Block (1), Log (0)]

FILE -> EXPORT -> PDF

Version Modified: NX6.0.1  
Scope: Session  
Type: Integer  
Default Name: UG\_IP\_menu\_export\_pdf  
Valid Options: [Block (1), Log (0)]

FILE -> EXPORT -> CGM

Version Modified: NX6.0.1  
Scope: Session  
Type: Integer  
Default Name: UG\_IP\_menu\_export\_cgm  
Valid Options: [Block (1), Log (0)]

FILE -> EXPORT -> STL

Version Modified: NX6.0.1  
Scope: Session  
Type: Integer  
Default Name: UG\_IP\_menu\_export\_stl  
Valid Options: [Block (1), Log (0)]

FILE -> EXPORT ->  
POLYGON FILE

Version Modified: NX6.0.1  
Scope: Session  
Type: Integer  
Default Name: UG\_IP\_menu\_export\_poly  
Valid Options: [Block (1), Log (0)]

FILE -> EXPORT ->  
AUTHOR HTML

Version Modified: NX6.0.1  
Scope: Session  
Type: Integer  
Default Name: UG\_IP\_menu\_export\_author  
Valid Options: [Block (1), Log (0)]



**FILE -> EXPORT -> JT**

Version Modified: NX6.0.1  
Scope: Session  
Type: Integer  
Default Name: UG\_IP\_menu\_export\_tcv  
Valid Options: [Block (1), Log (0)]

**FILE -> EXPORT ->  
VRML**

Version Modified: NX6.0.1  
Scope: Session  
Type: Integer  
Default Name: UG\_IP\_menu\_export\_vrml  
Valid Options: [Block (1), Log (0)]

**FILE -> EXPORT -> PNG**

Version Modified: NX6.0.1  
Scope: Session  
Type: Integer  
Default Name: UG\_IP\_menu\_export\_png  
Valid Options: [Block (1), Log (0)]

**FILE -> EXPORT ->  
JPEG AND MB3 ->  
CAPTURE IMAGE**

Version Modified: NX6.0.1  
Scope: Session  
Type: Integer  
Default Name: UG\_IP\_menu\_export\_jpeg  
Valid Options: [Block (1), Log (0)]

**FILE -> EXPORT -> GIF**

Version Modified: NX6.0.1  
Scope: Session  
Type: Integer  
Default Name: UG\_IP\_menu\_export\_gif  
Valid Options: [Block (1), Log (0)]

FILE -> EXPORT -> TIFF

Version Modified: NX6.0.1  
Scope: Session  
Type: Integer  
Default Name: UG\_IP\_menu\_export\_tiff  
Valid Options: [Block (1), Log (0)]

FILE -> EXPORT -> XWD

Version Modified: NX6.0.1  
Scope: Session  
Type: Integer  
Default Name: UG\_IP\_menu\_export\_xwd  
Valid Options: [Block (1), Log (0)]

FILE -> EXPORT -> BMP

Version Modified: NX6.0.1  
Scope: Session  
Type: Integer  
Default Name: UG\_IP\_menu\_export\_bmp  
Valid Options: [Block (1), Log (0)]

FILE -> EXPORT -> IGES

Version Modified: NX6.0.1  
Scope: Session  
Type: Integer  
Default Name: UG\_IP\_menu\_export\_iges  
Valid Options: [Block (1), Log (0)]

FILE -> EXPORT ->  
STEP203

Version Modified: NX6.0.1  
Scope: Session  
Type: Integer  
Default Name: UG\_IP\_menu\_export\_step203  
Valid Options: [Block (1), Log (0)]

**FILE -> EXPORT ->  
STEP214**

Version Modified: NX6.0.1  
Scope: Session  
Type: Integer  
Default Name: UG\_IP\_menu\_export\_step214  
Valid Options: [Block (1), Log (0)]

**FILE -> EXPORT ->  
DXF/DWG**

Version Modified: NX6.0.1  
Scope: Session  
Type: Integer  
Default Name: UG\_IP\_menu\_export\_dxf  
Valid Options: [Block (1), Log (0)]

**FILE -> EXPORT ->  
2D EXCHANGE**

Version Modified: NX6.0.1  
Scope: Session  
Type: Integer  
Default Name: UG\_IP\_menu\_export\_2d  
Valid Options: [Block (1), Log (0)]

**FILE -> EXPORT ->  
HEAL GEOMETRY**

Version Modified: NX6.0.1  
Scope: Session  
Type: Integer  
Default Name: UG\_IP\_menu\_export\_heal  
Valid Options: [Block (1), Log (0)]

**FILE -> EXPORT ->  
CATIA V4**

Version Modified: NX6.0.1  
Scope: Session  
Type: Integer  
Default Name: UG\_IP\_menu\_export\_v4catia

Valid Options: [Block (1), Log (0)]

FILE -> EXPORT ->  
CATIA V5

Version Modified: NX6.0.1

Scope: Session

Type: Integer

Default Name: UG\_IP\_menu\_export\_v5catia

Valid Options: [Block (1), Log (0)]

FILE -> INTEROPERATE

Version Modified: NX6.0.1

Scope: Session

Type: Integer

Default Name: UG\_IP\_menu\_file\_interoperate

Valid Options: [Block (1), Log (0)]

FILE -> COLLABORATE

Version Modified: NX6.0.1

Scope: Session

Type: Integer

Default Name: UG\_IP\_menu\_file\_collaborate

Valid Options: [Block (1), Log (0)]

FILE -> UTILITIES  
-> SAVE OUTSIDE  
TEAMCENTER

Version Modified: NX6.0.1

Scope: Session

Type: Integer

Default Name: UG\_IP\_menu\_file\_save\_outside

Valid Options: [Block (1), Log (0)]

EDIT -> COPY DISPLAY

Version Modified: NX6.0.1

Scope: Session

Type: Integer

Default Name: UG\_IP\_menu\_edit\_copy

Valid Options: [Block (1), Log (0)]

VIEW ->  
VISUALIZATION ->  
HIGH QUALITY IMAGE

Version Modified: NX6.0.1

Scope: Session

Type: Integer

Default Name: UG\_IP\_menu\_view\_vis\_high

Valid Options: [Block (1), Log (0)]

VIEW ->  
VISUALIZATION ->  
CREATE ANIMATION

Version Modified: NX6.0.1

Scope: Session

Type: Integer

Default Name: UG\_IP\_menu\_view\_vis\_create

Valid Options: [Block (1), Log (0)]

TOOLS -> MOVIE ->  
RECORD

Version Modified: NX6.0.1

Scope: Session

Type: Integer

Default Name: UG\_IP\_menu\_tools\_movie

Valid Options: [Block (1), Log (0)]

TOOLS -> PART  
FAMILIES

Version Modified: NX6.0.1

Scope: Session

Type: Integer

Default Name: UG\_IP\_menu\_tools\_partfamilies\_create

Valid Options: [Block (1), Log (0)]

ASSEMBLIES ->  
COMPONENTS -> ADD  
COMPONENT

Version Modified: NX6.0.1

Scope: Session

Type: Integer  
Default Name: UG\_IP\_menu\_assemblies\_comp\_add  
Valid Options: [Block (1), Log (0)]

ASSEMBLIES ->  
COMPONENTS  
-> CREATE NEW  
COMPONENT

Version Modified: NX6.0.1  
Scope: Session  
Type: Integer  
Default Name: UG\_IP\_menu\_assemblies\_comp\_create  
Valid Options: [Block (1), Log (0)]

ASSEMBLIES ->  
COMPONENTS ->  
CREATE NEW PARENT

Version Modified: NX6.0.1  
Scope: Session  
Type: Integer  
Default Name: UG\_IP\_menu\_assemblies\_comp\_parent  
Valid Options: [Block (1), Log (0)]

ASSEMBLIES ->  
COMPONENTS  
-> REPLACE  
COMPONENT

Version Modified: NX6.0.1  
Scope: Session  
Type: Integer  
Default Name: UG\_IP\_menu\_assemblies\_comp\_substitute  
Valid Options: [Block (1), Log (0)]

ASSEMBLIES ->  
COMPONENTS -> PART  
FAMILY UPDATE

Version Modified: NX6.0.1  
Scope: Session  
Type: Integer  
Default Name: UG\_IP\_menu\_assemblies\_comp\_part  
Valid Options: [Block (1), Log (0)]

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ASSEMBLIES ->  
CLONING -> CREATE  
CLONE ASSEMBLY

Version Modified: NX6.0.1  
Scope: Session  
Type: Integer  
Default Name: UG\_IP\_menu\_assemblies\_cloning\_create  
Valid Options: [Block (1), Log (0)]

ASSEMBLIES ->  
WAVE -> RELATIONS  
BROWSER

Version Modified: NX6.0.1  
Scope: Session  
Type: Integer  
Default Name: UG\_IP\_menu\_assemblies\_relations  
Valid Options: [Block (1), Log (0)]

SEQUENCING  
TOOLS -> EXPORT  
TO MOVIE

Version Modified: NX6.0.2  
Scope: Session  
Type: Integer  
Default Name: UG\_IP\_menu\_tools\_export\_to\_movie  
Valid Options: [Block (1), Log (0)]

SIMULATION  
NAVIGATOR  
MB3 -> EXPORT

Version Modified: NX6.0.2  
Scope: Session  
Type: Integer  
Default Name: UG\_IP\_menu\_field\_export  
Valid Options: [Block (1), Log (0)]

ADVANCED  
SIMULATION  
FILE -> EXPORT ->  
ANIMATED GIF

Version Modified: NX6.0.2

Scope: Session  
Type: Integer  
Default Name: UG\_IP\_menu\_file\_export\_animgif  
Valid Options: [Block (1), Log (0)]

FILE -> EXPORT ->  
SIMULATION

Version Modified: NX6.0.2  
Scope: Session  
Type: Integer  
Default Name: UG\_IP\_menu\_export\_sim  
Valid Options: [Block (1), Log (0)]

TOOLS -> POST ->  
EXPORT JT

Version Modified: NX6.0.2  
Scope: Session  
Type: Integer  
Default Name: UG\_IP\_menu\_tools\_post\_exportjt  
Valid Options: [Block (1), Log (0)]

TOOLS -> EXPORT  
REPORT

Version Modified: NX6.0.2  
Scope: Session  
Type: Integer  
Default Name: UG\_IP\_menu\_tools\_export\_report  
Valid Options: [Block (1), Log (0)]

MOTION SIMULATION  
FILE -> EXPORT ->  
RECURDYN INPUT

Version Modified: NX6.0.2  
Scope: Session  
Type: Integer  
Default Name: UG\_IP\_menu\_export\_recurdyn\_solver  
Valid Options: [Block (1), Log (0)]



**FILE -> EXPORT -> ADM**

Version Modified: NX6.0.2  
Scope: Session  
Type: Integer  
Default Name: UG\_IP\_menu\_export\_adm\_dataset  
Valid Options: [Block (1), Log (0)]

**FILE -> EXPORT ->  
ADAMS INPUT**

Version Modified: NX6.0.2  
Scope: Session  
Type: Integer  
Default Name: UG\_IP\_menu\_export\_adams\_solver  
Valid Options: [Block (1), Log (0)]

**FILE -> EXPORT ->  
EXPORT MECHANISM**

Version Modified: NX6.0.2  
Scope: Session  
Type: Integer  
Default Name: UG\_IP\_menu\_export\_mech  
Valid Options: [Block (1), Log (0)]

**FILE -> EXPORT ->  
MPEG**

Version Modified: NX6.0.2  
Scope: Session  
Type: Integer  
Default Name: UG\_IP\_menu\_export\_mpeg  
Valid Options: [Block (1), Log (0)]

**FILE -> EXPORT ->  
MPEG2**

Version Modified: NX6.0.2  
Scope: Session  
Type: Integer  
Default Name: UG\_IP\_menu\_export\_mpeg2  
Valid Options: [Block (1), Log (0)]

**FILE -> EXPORT ->  
ANIMATED GIF**

Version Modified: NX6.0.2  
Scope: Session  
Type: Integer  
Default Name: UG\_IP\_menu\_export\_anim\_gif  
Valid Options: [Block (1), Log (0)]

**FILE -> EXPORT ->  
ANIMATED TIFF**

Version Modified: NX6.0.2  
Scope: Session  
Type: Integer  
Default Name: UG\_IP\_menu\_export\_anim\_tiff  
Valid Options: [Block (1), Log (0)]

**FILE -> EXPORT  
-> TEAMCENTER  
VISUALIZATION**

Version Modified: NX6.0.2  
Scope: Session  
Type: Integer  
Default Name: UG\_IP\_menu\_export\_tce\_vis  
Valid Options: [Block (1), Log (0)]

---

**File New**

---

**ASSIGN ID  
AND REVISION  
AUTOMATICALLY**

This option determines whether or not the id and revision should be automatically generated.

Version Modified: NX5.0.5

Scope: Session

Type: Logical

Default Name: UGMGR\_AutoGenerateItemID

Valid Options: [Off (false), On (true)]

---

## Assembly

---

**IMPORT  
ASSEMBLY-LEVEL  
JT GEOMETRY  
OVERRIDES**

This option determines whether or not assembly-level JT geometry overrides are imported from Teamcenter into NX when a multi CAD assembly is loaded.

Version Modified: NX6.0.2

Scope: Session

Type: Logical

Default Name: UGMGR\_ImportJtOverrides

Valid Options: [Off (false), On (true)]

## Vehicle Design

---

## General Options

---

### ALLOW INPUT VALUES OUTSIDE STANDARD RANGE

This option determines whether or not to allow input values outside the standard range. If the option is on, the user can use any input values to investigate alternative

Version Modified: NX6.0.2  
Scope: Session  
Type: Logical  
Default Name: VD\_ALLOW\_NON\_STANDARD\_INPUT  
Valid Options: [On (true), Off (false)]

---

## Pedestrian Protection

---

### GENERAL USE MARGINAL VALUES

This option determines whether or not the marginal values should be used in the analysis or not.

Version Modified: NX6.0.1  
Scope: Session  
Type: Logical  
Default Name: VD\_PEDPRO\_USE\_MARGINAL  
Valid Options: [No (false), Yes (true)]

### EURO NCAP GRID PASS COLOR

Version Modified: NX6.0.1  
Scope: Session  
Type: Integer  
Default Name: VD\_PEDPRO\_Grid\_Pass\_Color  
Valid Options: [Default]

### WARNING COLOR

Version Modified: NX6.0.1  
Scope: Session  
Type: Integer  
Default Name: VD\_PEDPRO\_Grid\_Warning\_Color  
Valid Options: [Default]

### FAIL COLOR

Version Modified: NX6.0.1  
Scope: Session  
Type: Integer  
Default Name: VD\_PEDPRO\_Grid\_Fail\_Color  
Valid Options: [Default]

### FONT

Version Modified: NX6.0.1  
Scope: Session

Type: Integer  
 Default Name: VD\_PEDPRO\_Grid\_Font  
 Valid Options: [Default (2147483647), Solid (1), Dashed (2), Phantom (3), Centerline (4), Dotted (5), Long Dashed (6), Dotted Dashed (7)]

**WIDTH**

Version Modified: NX6.0.1  
 Scope: Session  
 Type: Integer  
 Default Name: VD\_PEDPRO\_Grid\_Width  
 Valid Options: [Default (2147483647), Thin (2), Normal (0), Thick (1)]

**LAYER**

Version Modified: NX6.0.1  
 Scope: Session  
 Type: Integer  
 Default Name: VD\_PEDPRO\_AdultFullGrid\_Layer  
 Valid Range: [ >=0 <=256 ]

**LAYER**

Version Modified: NX6.0.1  
 Scope: Session  
 Type: Integer  
 Default Name: VD\_PEDPRO\_AdultPartialGrid\_Layer  
 Valid Range: [ >=0 <=256 ]

**LAYER**

Version Modified: NX6.0.1  
 Scope: Session  
 Type: Integer  
 Default Name: VD\_PEDPRO\_ChildFullGrid\_Layer  
 Valid Range: [ >=0 <=256 ]

**LAYER**

Version Modified: NX6.0.1  
 Scope: Session

Type: Integer  
Default Name: VD\_PEDPRO\_ChildPartialGrid\_Layer  
Valid Range: [ >=0 <=256 ]

**LAYER**

Version Modified: NX6.0.1  
Scope: Session  
Type: Integer  
Default Name: VD\_PEDPRO\_UntranslatedGrid\_Layer  
Valid Range: [ >=0 <=256 ]

**GLOBAL TECHNICAL  
REGULATION  
STANDARDS  
MARGINAL  
HEADFORM DIAMETER**

This option specifies the diameter of the head form when using the marginal option.

Version Modified: NX6.0.1  
Scope: Session  
Type: Real  
Default Name: VD\_PEDPRO\_Head\_Diameter\_Marginal\_MU  
Applies To: Metric  
Units: mm

**MARGINAL  
HEADFORM DIAMETER**

This option specifies the diameter of the head form when using the marginal option.

Version Modified: NX6.0.1  
Scope: Session  
Type: Real  
Default Name: VD\_PEDPRO\_Head\_Diameter\_Marginal\_EU  
Applies To: English  
Units: in

**MARGINAL FRONT  
ANGLE**

This option specifies the front reference angle when using the marginal option.

Version Modified: NX6.0.1



Scope: Session  
 Type: Real  
 Default Name: VD\_PEDPRO\_Front\_Reference\_Angle\_Marginal  
 Units: deg

**EURO NCAP FRONT  
 ANGLE**

This option specifies the front reference angle for the Euro NCAP standard.

Version Modified: NX6.0.1  
 Scope: Session  
 Type: Real  
 Default Name: VD\_PEDPRO\_Front\_Reference\_Angle\_NCAP  
 Units: deg

**EURO NCAP  
 MARGINAL FRONT  
 ANGLE**

This option specifies the front reference angle for the Euro NCAP standard when using the marginal option.

Version Modified: NX6.0.1  
 Scope: Session  
 Type: Real  
 Default Name: VD\_PEDPRO\_Front\_Reference\_Angle\_NCAP\_Marginal  
 Units: deg

**MARGINAL SIDE  
 ANGLE**

This option specifies the side reference angle when using the marginal option.

Version Modified: NX6.0.1  
 Scope: Session  
 Type: Real  
 Default Name: VD\_PEDPRO\_Side\_Reference\_Angle\_Marginal  
 Units: deg

**EURO NCAP SIDE  
 ANGLE**

This option specifies the side reference angle for the Euro NCAP standard.

Version Modified: NX6.0.1  
 Scope: Session  
 Type: Real

Default Name: VD\_PEDPRO\_Side\_Reference\_Angle\_NCAP

Units: deg

**EURO NCAP  
MARGINAL SIDE  
ANGLE**

This option specifies the side reference angle for the Euro NCAP standard when using the marginal option.

Version Modified: NX6.0.1

Scope: Session

Type: Real

Default Name: VD\_PEDPRO\_Side\_Reference\_Angle\_NCAP\_Marginal

Units: deg

**MARGINAL CORNER  
(CA)**

This option specifies the corner bumper angle for the leg impact zone when using the marginal option.

Version Modified: NX6.0.1

Scope: Session

Type: Real

Default Name: VD\_PEDPRO\_Corner\_Angle\_Marginal

Units: deg

**WAD LINE VALUES**

This option specifies the wad lines that are to be allowed.

Version Modified: NX6.0.1

Scope: Session

Type: String

Default Name: VD\_PEDPRO\_WAD\_MU

Applies To: Metric

Units: mm

**WAD LINE VALUES**

This option specifies the wad lines that are to be allowed.

Version Modified: NX6.0.1

Scope: Session

Type: String

Default Name: VD\_PEDPRO\_WAD\_EU

Applies To: English

Units: in

**MARGINAL WAD LINE  
VALUES**

This option specifies the wad lines that are to be allowed when using the marginal option.

Version Modified: NX6.0.1

Scope: Session

Type: String

Default Name: VD\_PEDPRO\_MARGINAL\_WAD\_MU

Applies To: Metric

Units: mm

**MARGINAL WAD LINE  
VALUES**

This option specifies the wad lines that are to be allowed when using the marginal option.

Version Modified: NX6.0.1

Scope: Session

Type: String

Default Name: VD\_PEDPRO\_MARGINAL\_WAD\_EU

Applies To: English

Units: in

**EURO NCAP WAD  
LINE VALUES**

This option specifies the wad lines that are to be used in creating the Euro NCAP grid.

Version Modified: NX6.0.1

Scope: Session

Type: String

Default Name: VD\_PEDPRO\_NCAP\_WAD\_MU

Applies To: Metric

Units: mm

**EURO NCAP WAD  
LINE VALUES**

This option specifies the wad lines that are to be used in creating the Euro NCAP grid.

Version Modified: NX6.0.1  
Scope: Session  
Type: String  
Default Name: VD\_PEDPRO\_NCAP\_WAD\_EU  
Applies To: English  
Units: in

**EURO NCAP  
MARGINAL WAD  
LINE VALUES**

This option specifies the wad lines that are to be used in creating the Euro NCAP grid when using the marginal option.

Version Modified: NX6.0.1  
Scope: Session  
Type: String  
Default Name: VD\_PEDPRO\_NCAP\_MARGINAL\_WAD\_MU  
Applies To: Metric  
Units: mm

**EURO NCAP  
MARGINAL WAD  
LINE VALUES**

This option specifies the wad lines that are to be used in creating the Euro NCAP grid when using the marginal option.

Version Modified: NX6.0.1  
Scope: Session  
Type: String  
Default Name: VD\_PEDPRO\_NCAP\_MARGINAL\_WAD\_EU  
Applies To: English  
Units: in

**EUROPEAN  
STANDARDS  
MARGINAL  
HEADFORM DIAMETER**

This option specifies the diameter of the head form when using the marginal option.

Version Modified: NX6.0.1  
Scope: Session  
Type: Real

Default Name: VD\_PEDPRO\_EURO\_Head\_Diameter\_Marginal\_MU  
 Applies To: English  
 Units: mm

**MARGINAL  
 HEADFORM DIAMETER**

This option specifies the diameter of the head form when using the marginal option.

Version Modified: NX6.0.1

Scope: Session

Type: Real

Default Name: VD\_PEDPRO\_EURO\_Head\_Diameter\_Marginal\_EU

Applies To: English

Units: in

**MARGINAL FRONT  
 ANGLE**

This option specifies the front reference angle when using the marginal option.

Version Modified: NX6.0.1

Scope: Session

Type: Real

Default Name: VD\_PEDPRO\_EURO\_Front\_Reference\_Angle\_Marginal

Applies To: English

Units: deg

**EURO NCAP FRONT  
 ANGLE**

This option specifies the front reference angle for the Euro NCAP standard.

Version Modified: NX6.0.1

Scope: Session

Type: Real

Default Name: VD\_PEDPRO\_EURO\_Front\_Reference\_Angle\_NCAP

Applies To: English

Units: deg

**EURO NCAP  
 MARGINAL FRONT  
 ANGLE**

This option specifies the front reference angle for the Euro NCAP standard when using the marginal option.

Version Modified: NX6.0.1  
Scope: Session  
Type: Real  
Default Name: VD\_PEDPRO\_EURO\_Front\_Reference\_Angle\_NCAP\_Marginal  
Applies To: English  
Units: deg

**MARGINAL SIDE  
ANGLE**

This option specifies the side reference angle when using the marginal option.

Version Modified: NX6.0.1  
Scope: Session  
Type: Real  
Default Name: VD\_PEDPRO\_EURO\_Side\_Reference\_Angle\_Marginal  
Applies To: English  
Units: deg

**EURO NCAP SIDE  
ANGLE**

This option specifies the side reference angle for the Euro NCAP standard.

Version Modified: NX6.0.1  
Scope: Session  
Type: Real  
Default Name: VD\_PEDPRO\_EURO\_Side\_Reference\_Angle\_NCAP  
Applies To: English  
Units: deg

**EURO NCAP  
MARGINAL SIDE  
ANGLE**

This option specifies the side reference angle for the Euro NCAP standard when using the marginal option.

Version Modified: NX6.0.1  
Scope: Session  
Type: Real  
Default Name: VD\_PEDPRO\_EURO\_Side\_Reference\_Angle\_NCAP\_Marginal  
Applies To: English  
Units: deg

**MARGINAL CORNER  
(CA)**

This option specifies the corner bumper angle for the leg impact zone when using the marginal option.

Version Modified: NX6.0.1  
Scope: Session  
Type: Real  
Default Name: VD\_PEDPRO\_EURO\_Corner\_Angle\_Marginal  
Applies To: English  
Units: deg

**WAD LINE VALUES**

This option specifies the wad lines that are to be allowed.

Version Modified: NX6.0.1  
Scope: Session  
Type: String  
Default Name: VD\_PEDPRO\_EURO\_WAD\_MU  
Applies To: English  
Units: mm

**WAD LINE VALUES**

This option specifies the wad lines that are to be allowed.

Version Modified: NX6.0.1  
Scope: Session  
Type: String  
Default Name: VD\_PEDPRO\_EURO\_WAD\_EU  
Applies To: English  
Units: in

**MARGINAL WAD LINE  
VALUES**

This option specifies the wad lines that are to be allowed when using the marginal option.

Version Modified: NX6.0.1  
Scope: Session  
Type: String  
Default Name: VD\_PEDPRO\_EURO\_MARGINAL\_WAD\_MU

Applies To: English

Units: mm

**MARGINAL WAD LINE  
VALUES**

This option specifies the wad lines that are to be allowed when using the marginal option.

Version Modified: NX6.0.1

Scope: Session

Type: String

Default Name: VD\_PEDPRO\_EURO\_MARGINAL\_WAD\_EU

Applies To: English

Units: in

**EURO NCAP WAD  
LINE VALUES**

This option specifies the wad lines that are to be used in creating the Euro NCAP grid.

Version Modified: NX6.0.1

Scope: Session

Type: String

Default Name: VD\_PEDPRO\_EURO\_NCAP\_WAD\_MU

Applies To: English

Units: mm

**EURO NCAP WAD  
LINE VALUES**

This option specifies the wad lines that are to be used in creating the Euro NCAP grid.

Version Modified: NX6.0.1

Scope: Session

Type: String

Default Name: VD\_PEDPRO\_EURO\_NCAP\_WAD\_EU

Applies To: English

Units: in

**EURO NCAP  
MARGINAL WAD  
LINE VALUES**

This option specifies the wad lines that are to be used in creating the Euro NCAP grid when using the marginal option.



Version Modified: NX6.0.1  
 Scope: Session  
 Type: String  
 Default Name: VD\_PEDPRO\_EURO\_NCAP\_MARGINAL\_WAD\_MU  
 Applies To: English  
 Units: mm

**EURO NCAP  
 MARGINAL WAD  
 LINE VALUES**

This option specifies the wad lines that are to be used in creating the Euro NCAP grid when using the marginal option.

Version Modified: NX6.0.1  
 Scope: Session  
 Type: String  
 Default Name: VD\_PEDPRO\_EURO\_NCAP\_MARGINAL\_WAD\_EU  
 Applies To: English  
 Units: in

**JAPANESE  
 STANDARDS  
 MARGINAL  
 HEADFORM DIAMETER**

This option specifies the diameter of the head form when using the marginal option.

Version Modified: NX6.0.1  
 Scope: Session  
 Type: Real  
 Default Name: VD\_PEDPRO\_JAPAN\_Head\_Diameter\_Marginal\_MU  
 Applies To: Metric  
 Units: mm

**MARGINAL  
 HEADFORM DIAMETER**

This option specifies the diameter of the head form when using the marginal option.

Version Modified: NX6.0.1  
 Scope: Session  
 Type: Real

Default Name: VD\_PEDPRO\_JAPAN\_Head\_Diameter\_Marginal\_EU  
Applies To: English  
Units: in

**MARGINAL FRONT  
ANGLE**

This option specifies the front reference angle when using the marginal option.

Version Modified: NX6.0.1

Scope: Session

Type: Real

Default Name: VD\_PEDPRO\_JAPAN\_Front\_Reference\_Angle\_Marginal

Units: deg

**EURO NCAP FRONT  
ANGLE**

This option specifies the front reference angle for the Euro NCAP standard.

Version Modified: NX6.0.1

Scope: Session

Type: Real

Default Name: VD\_PEDPRO\_JAPAN\_Front\_Reference\_Angle\_NCAP

Units: deg

**EURO NCAP  
MARGINAL FRONT  
ANGLE**

This option specifies the front reference angle for the Euro NCAP standard when using the marginal option.

Version Modified: NX6.0.1

Scope: Session

Type: Real

Default Name: VD\_PEDPRO\_JAPAN\_Front\_Reference\_Angle\_NCAP\_Marginal

Units: deg

**MARGINAL SIDE  
ANGLE**

This option specifies the side reference angle when using the marginal option.

Version Modified: NX6.0.1

Scope: Session

Type: Real

Default Name: VD\_PEDPRO\_JAPAN\_Side\_Reference\_Angle\_Marginal

Units: deg

**EURO NCAP SIDE  
ANGLE**

This option specifies the side reference angle for the Euro NCAP standard.

Version Modified: NX6.0.1

Scope: Session

Type: Real

Default Name: VD\_PEDPRO\_JAPAN\_Side\_Reference\_Angle\_NCAP

Units: deg

**EURO NCAP  
MARGINAL SIDE  
ANGLE**

This option specifies the side reference angle for the Euro NCAP standard when using the marginal option.

Version Modified: NX6.0.1

Scope: Session

Type: Real

Default Name: VD\_PEDPRO\_JAPAN\_Side\_Reference\_Angle\_NCAP\_Marginal

Units: deg

**MARGINAL CORNER  
(CA)**

This option specifies the corner bumper angle for the leg impact zone when using the marginal option.

Version Modified: NX6.0.1

Scope: Session

Type: Real

Default Name: VD\_PEDPRO\_JAPAN\_Corner\_Angle\_Marginal

Units: deg

**WAD LINE VALUES**

This option specifies the wad lines that are to be allowed.

Version Modified: NX6.0.1

Scope: Session

Type: String

Default Name: VD\_PEDPRO\_JAPAN\_WAD\_MU

Applies To: Metric

Units: mm

**WAD LINE VALUES**

This option specifies the wad lines that are to be allowed.

Version Modified: NX6.0.1  
Scope: Session  
Type: String  
Default Name: VD\_PEDPRO\_JAPAN\_WAD\_EU  
Applies To: English  
Units: in

**MARGINAL WAD LINE  
VALUES**

This option specifies the wad lines that are to be allowed when using the marginal option.

Version Modified: NX6.0.1  
Scope: Session  
Type: String  
Default Name: VD\_PEDPRO\_JAPAN\_MARGINAL\_WAD\_MU  
Applies To: Metric  
Units: mm

**MARGINAL WAD LINE  
VALUES**

This option specifies the wad lines that are to be allowed when using the marginal option.

Version Modified: NX6.0.1  
Scope: Session  
Type: String  
Default Name: VD\_PEDPRO\_JAPAN\_MARGINAL\_WAD\_EU  
Applies To: English  
Units: in

**EURO NCAP WAD  
LINE VALUES**

This option specifies the wad lines that are to be used in creating the Euro NCAP grid.

Version Modified: NX6.0.1  
Scope: Session  
Type: String

Default Name: VD\_PEDPRO\_JAPAN\_NCAP\_WAD\_MU  
 Applies To: Metric  
 Units: mm

**EURO NCAP WAD  
 LINE VALUES**

This option specifies the wad lines that are to be used in creating the Euro NCAP grid.

Version Modified: NX6.0.1  
 Scope: Session  
 Type: String  
 Default Name: VD\_PEDPRO\_JAPAN\_NCAP\_WAD\_EU  
 Applies To: English  
 Units: in

**EURO NCAP  
 MARGINAL WAD  
 LINE VALUES**

This option specifies the wad lines that are to be used in creating the Euro NCAP grid when using the marginal option.

Version Modified: NX6.0.1  
 Scope: Session  
 Type: String  
 Default Name: VD\_PEDPRO\_JAPAN\_NCAP\_MARGINAL\_WAD\_MU  
 Applies To: Metric  
 Units: mm

**EURO NCAP  
 MARGINAL WAD  
 LINE VALUES**

This option specifies the wad lines that are to be used in creating the Euro NCAP grid when using the marginal option.

Version Modified: NX6.0.1  
 Scope: Session  
 Type: String  
 Default Name: VD\_PEDPRO\_JAPAN\_NCAP\_MARGINAL\_WAD\_EU  
 Applies To: English  
 Units: in

**KOREAN STANDARDS  
MARGINAL  
HEADFORM DIAMETER**

This option specifies the diameter of the head form when using the marginal option.

Version Modified: NX6.0.1

Scope: Session

Type: Real

Default Name: VD\_PEDPRO\_KOREAN\_Head\_Diameter\_Marginal\_MU

Applies To: Metric

Units: mm

**MARGINAL  
HEADFORM DIAMETER**

This option specifies the diameter of the head form when using the marginal option.

Version Modified: NX6.0.1

Scope: Session

Type: Real

Default Name: VD\_PEDPRO\_KOREAN\_Head\_Diameter\_Marginal\_EU

Applies To: English

Units: in

**MARGINAL FRONT  
ANGLE**

This option specifies the front reference angle when using the marginal option.

Version Modified: NX6.0.1

Scope: Session

Type: Real

Default Name: VD\_PEDPRO\_KOREAN\_Front\_Reference\_Angle\_Marginal

Units: deg

**EURO NCAP FRONT  
REFERENCE ANGLE**

This option specifies the front reference angle for the Euro NCAP standard.

Version Modified: NX6.0.1

Scope: Session

Type: Real

Default Name: VD\_PEDPRO\_KOREAN\_Front\_Reference\_Angle\_NCAP

Units: deg

**EURO NCAP  
MARGINAL FRONT  
ANGLE**

This option specifies the front reference angle for the Euro NCAP standard when using the marginal option.

Version Modified: NX6.0.1

Scope: Session

Type: Real

Default Name: VD\_PEDPRO\_KOREAN\_Front\_Reference\_Angle\_NCAP\_Marginal

Units: deg

**MARGINAL SIDE  
ANGLE**

This option specifies the side reference angle when using the marginal option.

Version Modified: NX6.0.1

Scope: Session

Type: Real

Default Name: VD\_PEDPRO\_KOREAN\_Side\_Reference\_Angle\_Marginal

Units: deg

**EURO NCAP SIDE  
ANGLE**

This option specifies the side reference angle for the Euro NCAP standard.

Version Modified: NX6.0.1

Scope: Session

Type: Real

Default Name: VD\_PEDPRO\_KOREAN\_Side\_Reference\_Angle\_NCAP

Units: deg

**EURO NCAP  
MARGINAL SIDE  
ANGLE**

This option specifies the side reference angle for the Euro NCAP standard when using the marginal option.

Version Modified: NX6.0.1

Scope: Session

Type: Real

Default Name: VD\_PEDPRO\_KOREAN\_Side\_Reference\_Angle\_NCAP\_Marginal

Units: deg

**MARGINAL CORNER  
(CA)**

This option specifies the corner bumper angle for the leg impact zone when using the marginal option.

Version Modified: NX6.0.1  
Scope: Session  
Type: Real  
Default Name: VD\_PEDPRO\_KOREAN\_Corner\_Angle\_Marginal  
Units: deg

**WAD LINE VALUES**

This option specifies the wad lines that are to be allowed.

Version Modified: NX6.0.1  
Scope: Session  
Type: String  
Default Name: VD\_PEDPRO\_KOREAN\_WAD\_MU  
Applies To: Metric  
Units: mm

**WAD LINE VALUES**

This option specifies the wad lines that are to be allowed.

Version Modified: NX6.0.1  
Scope: Session  
Type: String  
Default Name: VD\_PEDPRO\_KOREAN\_WAD\_EU  
Applies To: English  
Units: in

**MARGINAL WAD LINE  
VALUES**

This option specifies the wad lines that are to be allowed when using the marginal option.

Version Modified: NX6.0.1  
Scope: Session  
Type: String  
Default Name: VD\_PEDPRO\_KOREAN\_MARGINAL\_WAD\_MU  
Applies To: Metric



Units: mm

**MARGINAL WAD LINE  
VALUES**

This option specifies the wad lines that are to be allowed when using the marginal option.

Version Modified: NX6.0.1

Scope: Session

Type: String

Default Name: VD\_PEDPRO\_KOREAN\_MARGINAL\_WAD\_EU

Applies To: English

Units: in

**EURO NCAP WAD  
LINE VALUES**

This option specifies the wad lines that are to be used in creating the Euro NCAP grid.

Version Modified: NX6.0.1

Scope: Session

Type: String

Default Name: VD\_PEDPRO\_KOREAN\_NCAP\_WAD\_MU

Applies To: Metric

Units: mm

**EURO NCAP WAD  
LINE VALUES**

This option specifies the wad lines that are to be used in creating the Euro NCAP grid.

Version Modified: NX6.0.1

Scope: Session

Type: String

Default Name: VD\_PEDPRO\_KOREAN\_NCAP\_WAD\_EU

Applies To: English

Units: in

**EURO NCAP  
MARGINAL WAD  
LINE VALUES**

This option specifies the wad lines that are to be used in creating the Euro NCAP grid when using the marginal option.

Version Modified: NX6.0.1

Scope: Session

Type: String

Default Name: VD\_PEDPRO\_KOREAN\_NCAP\_MARGINAL\_WAD\_MU

Applies To: Metric

Units: mm

**EURO NCAP  
MARGINAL WAD  
LINE VALUES**

This option specifies the wad lines that are to be used in creating the Euro NCAP grid when using the marginal option.

Version Modified: NX6.0.1

Scope: Session

Type: String

Default Name: VD\_PEDPRO\_KOREAN\_NCAP\_MARGINAL\_WAD\_EU

Applies To: English

Units: in

## **Weld Assistant**

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## Weld Advisor

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### OUTER PANEL THICKNESS RATIO

This option specifies the thickness ratio that any two panels must not exceed. For example, if joining three panels of thickness 1, 2, and 4, the maximum thickness ratio is 4.0 (4 to 1).

Version Modified: NX5.0.5

Scope: Part

Type: Real

Default Name: Weld\_Advisor\_ThicknessOuterRatio

Valid Range: [ >0.0 ]