

PLMJobManagerV2

Refile – Dokumentation - Notes

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Inhaltsverzeichnis



Anwenderstandards Nx7.5

■ Einstellungen der Anwenderstandards

Manage Current Settings

Defaults Level: User

Application	Category	Tab	Setting	A...	Version Modified	Value	L...	Modified	Co...	Scope
Gateway	JT Files	Import/Export	Export Options - Save JT Data		NX3.0.0	No		Pending		Session
Gateway	JT Files	Teamcenter Integ...	Update JT Derived Parts from Source in Current Revision		NX4.0.1	No		Pending		Session
Teamcenter Integration for NX	General	General	Use Item Name instead of Item Number		NX6.0.0	No		Pending		Session
Teamcenter Integration for NX	General	Assembly	Structure Update on Load		NX4.0.0	Complete		Pending		Session
Teamcenter Integration for NX	General	Assembly	Structure Update on Save		NX3.0.0	No		Pending		Session
Teamcenter Integration for NX	General	Feature	Weld Points		NX3.0.0	No		Pending		Session
Teamcenter Integration for NX	General	Feature	Datum Points		NX3.0.0	No		Pending		Session
Teamcenter Integration for NX	General	Feature	Routing Data		NX4.0.0	No		Pending		Session
Teamcenter Integration for NX	General	Feature	Product Interfaces		NX5.0.0	No		Pending		Session

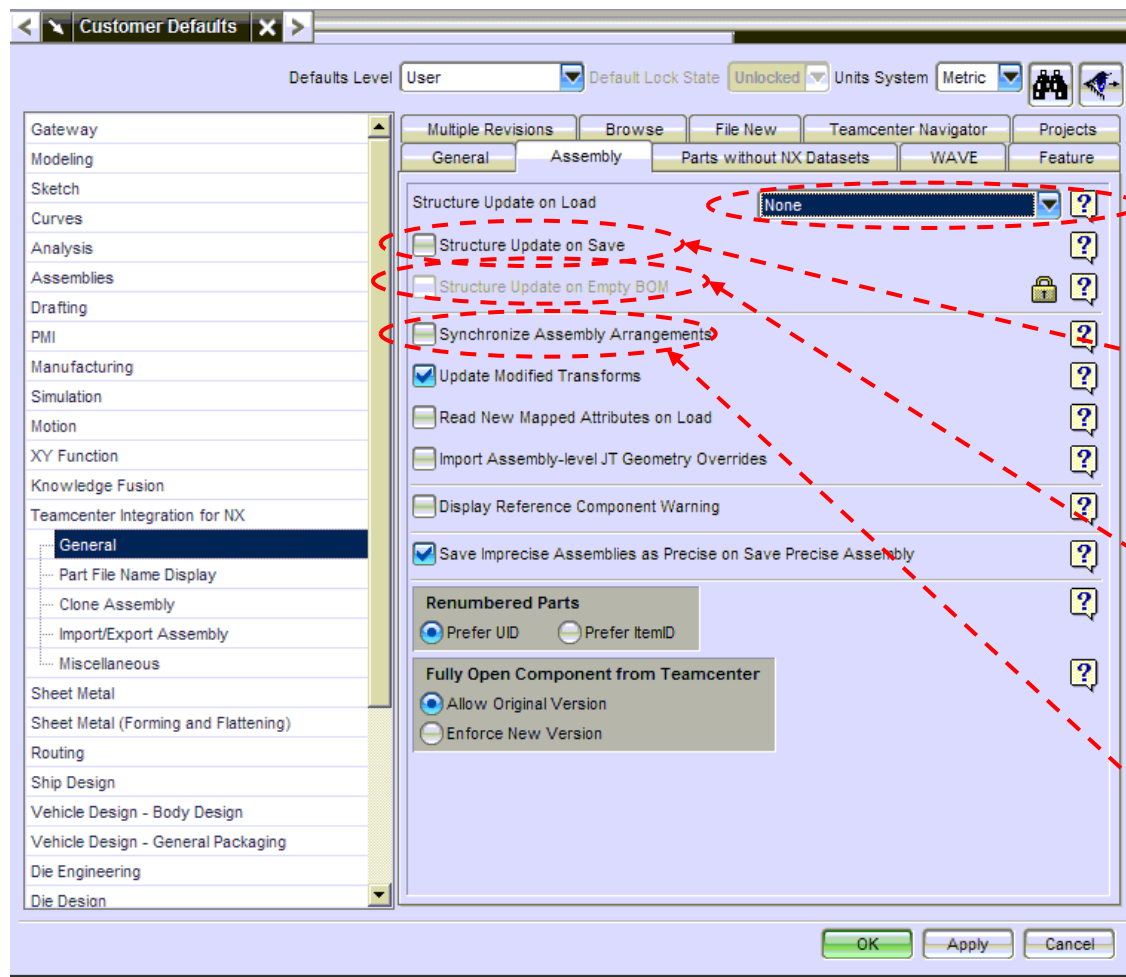
Description:

This option determines whether or not a JT file should be saved when parts are saved.

OK Cancel

Anwenderstandards Nx7.5

■ Einstellungen der Anwenderstandards



Enter words associated with default

UGMGR_StructureUpdateOnLoad

Find

Defaults Found:

Application	Category	Tab	Setting	Applies To	Version Modified
Teamcenter Integ...	General	Assembly	Structure Update...		NX4.0.0

Enter words associated with default

UGMGR_StructureUpdateOnSave

Find

Defaults Found:

Application	Category	Tab	Setting	Applies To	Version Modified
Teamcenter Integ...	General	Assembly	Structure Update...		NX3.0.0

Enter words associated with default

UGMGR_StructureUpdateOnEmptyBom

Find

Defaults Found:

Application	Category	Tab	Setting	Applies To	Version Modified
Teamcenter Integ...	General	Assembly	Structure Update...		NX6.0.0

Enter words associated with default

UGMGR_SyncArrangements

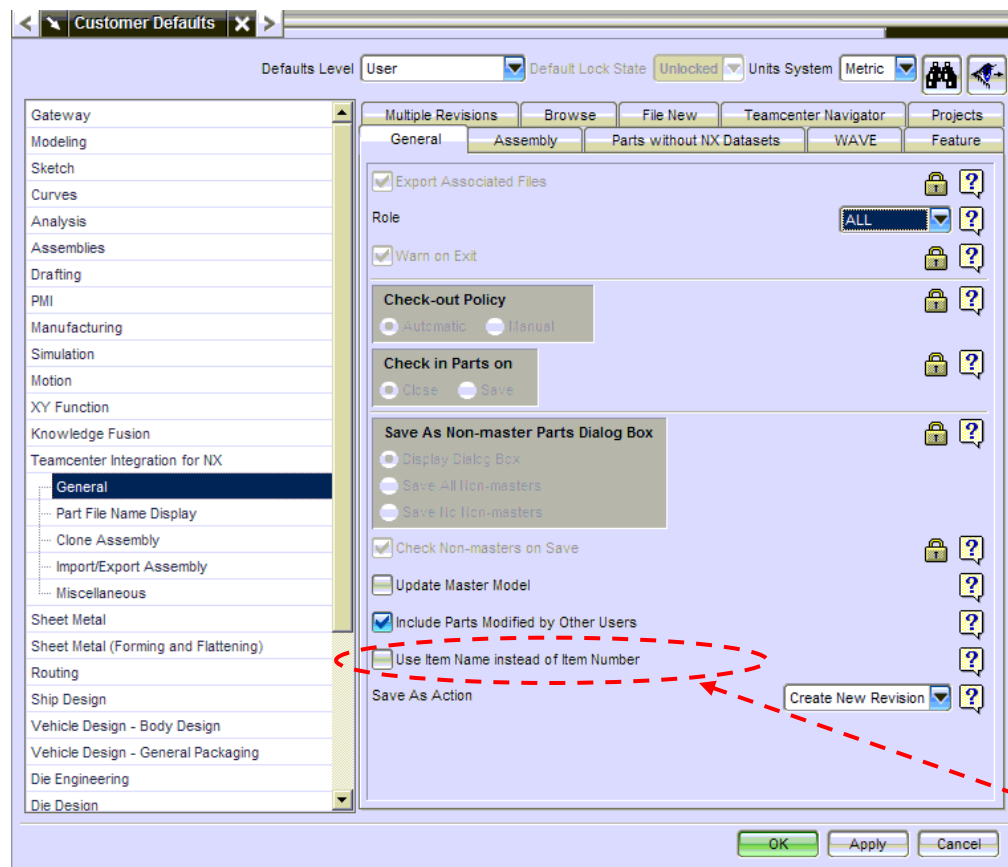
Find

Defaults Found:

Application	Category	Tab	Setting	Applies To	Version Modified
Teamcenter Integ...	General	Assembly	Synchronize As...		NX4.0.0

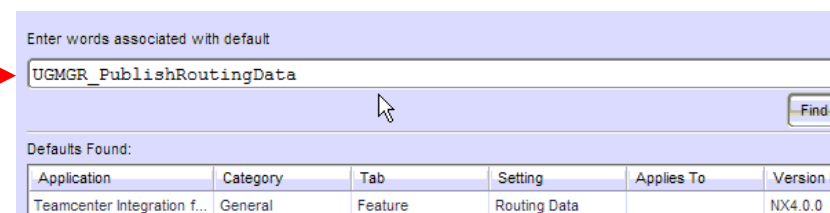
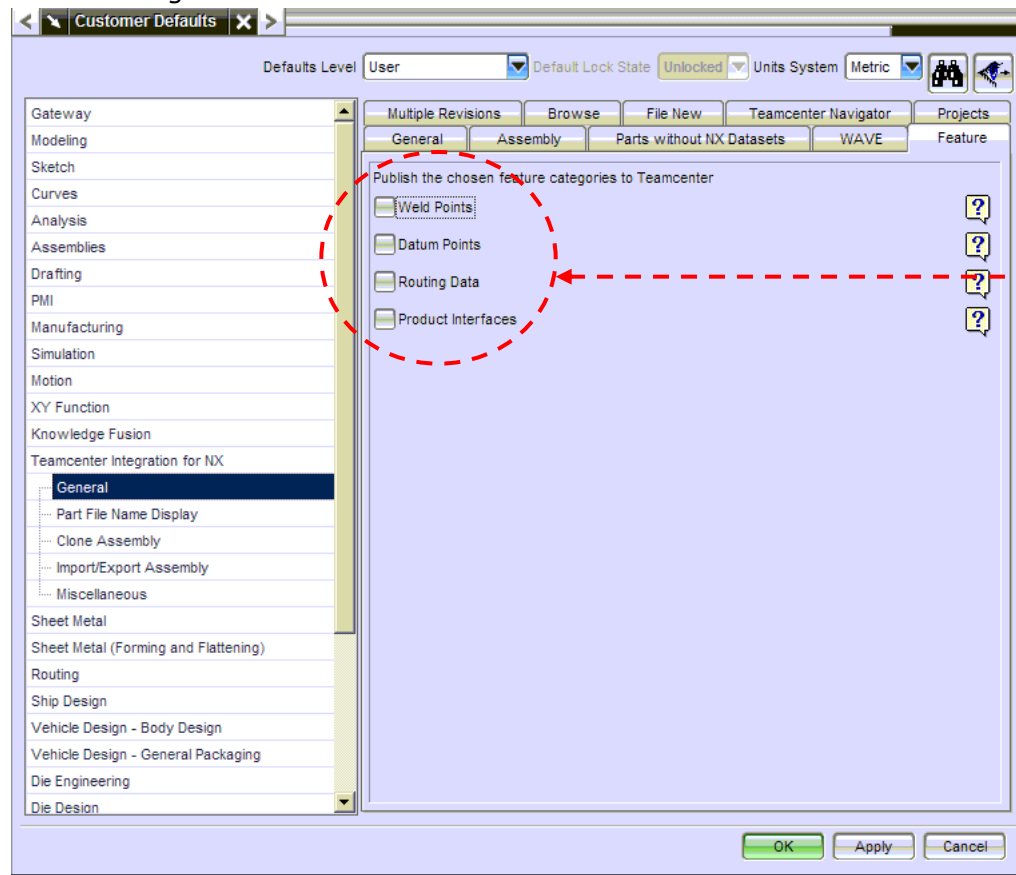
Anwenderstandards Nx7.5

■ Einstellungen der Anwenderstandards



Anwenderstandards Nx7.5

■ Einstellungen der Anwenderstandards



UGMGR_PublishWeldPoints
 UGMGR_PublishDatumPoints
 UGMGR_PublishRoutingData
 UGMGR_PublishProductInterfaces

Anwenderstandards Nx7.5

■ Einstellungen der Anwenderstandards

Customer Defaults

Defaults Level: User Default Lock State: Unlocked Units System: Metric

Import/Export | Extract Exact Data | Teamcenter Integration

Import Options

- Wireframe Layer: 1
- Reference Geometry Layer: 1
- ☒ Use Work Layer
- Model Geometry Layer: 1

Export Options

- ☐ Save JT Data

Teamcenter Integration

- ☐ Update JT Derived Parts from Source in Current Revision
- ☐ Delete Source JT on Save

Callout 1: Save JT Data

Enter words associated with default: Assemblies_SaveTeamCenterVisData

Defaults Found:

Application	Category	Tab	Setting	Applies To	Version Modified
Gateway	JT Files	Import/Export	Export Options - ...		NX3.0.0

Callout 2: Update JT Derived Parts from Source in Current Revision

Enter words associated with default: UGMGR_ForeignAutoRelinkJT

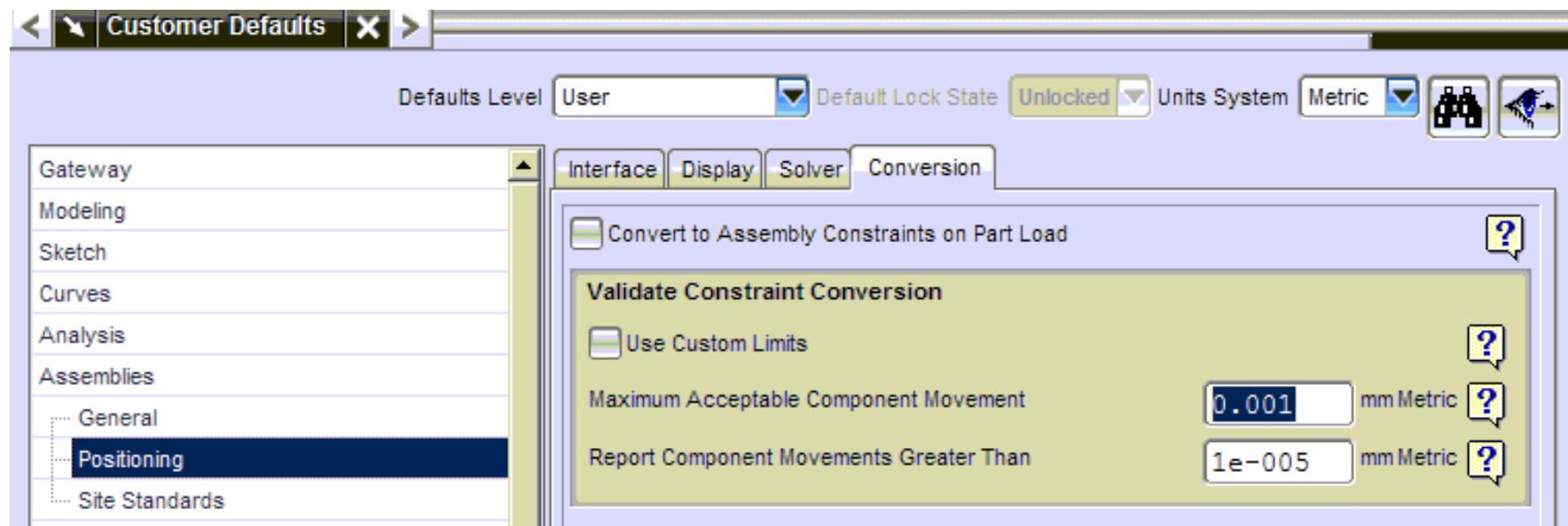
Defaults Found:

Application	Category	Tab	Setting	Applies To	Version Modified
Gateway	JT Files	Teamcenter Integ...	Update JT Derive...		NX4.0.1



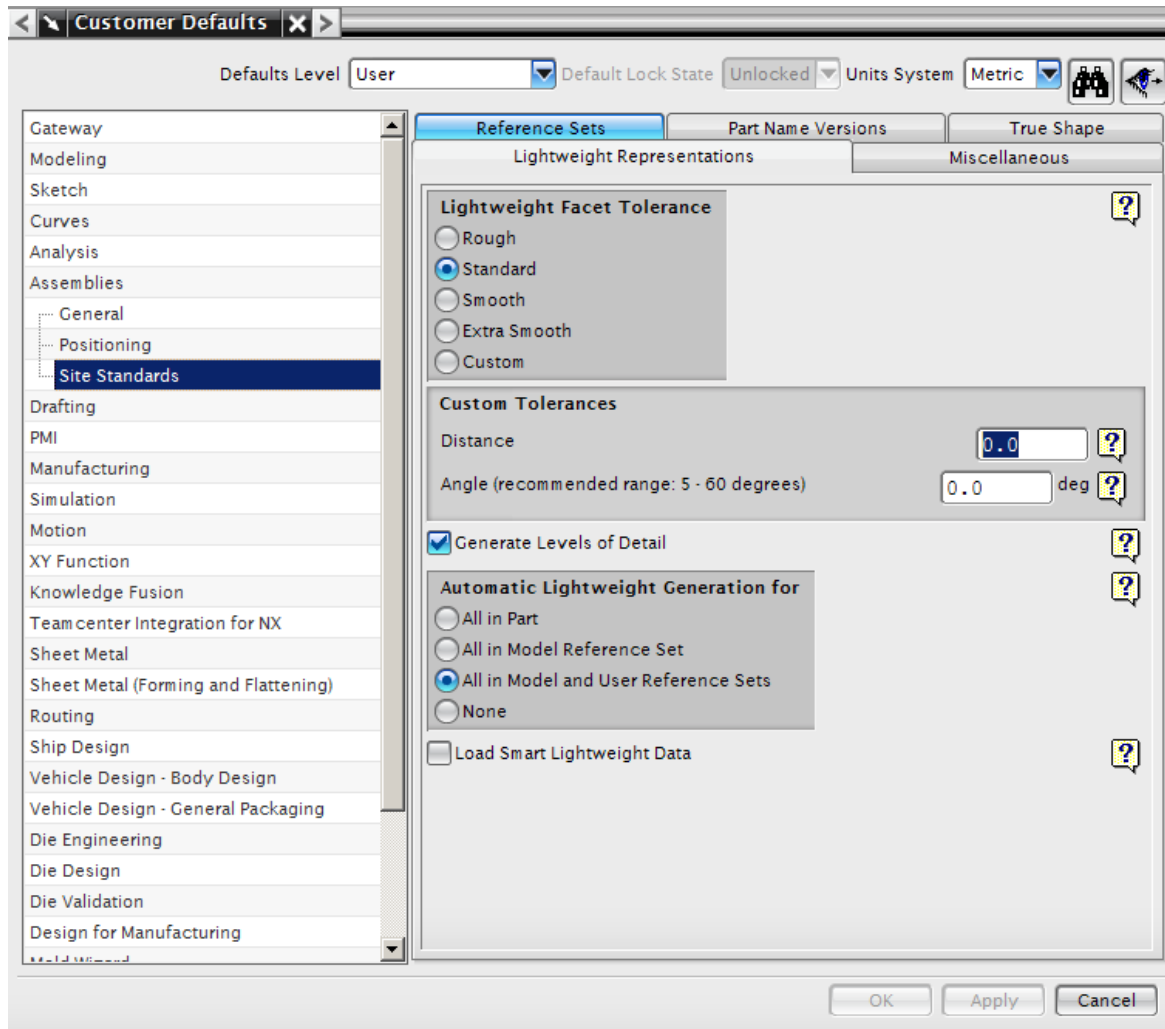
Anwenderstandards Nx7.5

■ Assemblies Positioning Conversion



Anwenderstandards Nx7.5

■ Lightweight Representation



NX 6 Baugruppen

■ Baugruppen Infos zum Thema „Mating Conditons“ ↔ „Assembly Constraints“

The screenshot displays the NX 6 software interface with the 'Anwenderstandards' (User Standards) dialog box open. The 'Interface' tab is selected, showing the 'Assembly Constraints' section. The 'Positioning' radio button is selected, and the 'Interpart Copy' checkbox is unchecked. The 'Constraint Conversion Validation' section is also visible, with the 'Use Custom Limits' checkbox checked. The 'Maximum acceptable component movement' is set to 0.001 mm Metrisch, and the 'Report component movements greater than' is set to 1e-005 mm Metrisch.

On the right side, the 'Baugruppen-Navigator' (Assembly Navigator) is open, showing a list of assembly components. The 'Vorschau' (Preview) pane shows 'Keine Vorschau vorhanden' (No preview available). The 'Abhängigkeiten' (Dependencies) pane shows a tree structure of the assembly.

Below the main dialog, the 'Customer Defaults' dialog box is open, showing the 'Conversion' tab. The 'Convert to Assembly Constraints on Part Load' checkbox is checked. The 'Validate Constraint Conversion' section is also visible, with the 'Use Custom Limits' checkbox checked. The 'Maximum Acceptable Component Movement' is set to 0.001 mm Metric, and the 'Report Component Movements Greater Than' is set to 1e-005 mm Metric.

Exact lightweight geometry and refile_part utility changes

Exact lightweight geometry and refile_part utility changes

What is it?

The lightweight (faceted) representation format has been enhanced to contain exact surface geometry information for faces with analytic surface geometry such as faces with planar, cylindrical, spherical, or toroidal surface geometry. The software uses the exact geometry information while performing certain operations on faces, edges, and vertices of lightweight bodies. This information enables the software to perform the operations on lightweight bodies with the same accuracy as on solids. Examples of operations where exact geometry information is used include **Move Component** and many types of measurement.

What's New in NX 7 and 7.5 9-27

Gateway

If the exact geometry in the part is created or updated in NX 6.0.2 or later, then the exact geometry information will be included in the lightweight representation and used by NX where possible. For parts with geometry last modified prior to NX 6.0.2, you must regenerate the lightweight representations to benefit from the improvements.

The refile_part and ugmanager_refile utilities have been enhanced to facilitate the regeneration.

Switch Description

regen_lw Regenerates all lightweight representations in the part, in order to take advantage of NX 6.0.2

enhancements to the lightweight format, such as the embedding of exact surface geometry definitions for faces with analytic geometry. regen_lw_def_tol Regenerates all lightweight bodies using the current default faceting tolerance values.

Note

In Teamcenter Integration, regen_lw_def_tol must be used in conjunction with regen_lw to take effect.

Why should I use it?

These enhancements enable you to get precise results in some important situations where you would previously have gotten approximate results due to the faceted nature of the representations.

Where do I find it?

Lightweight representations created or edited in NX 6.0.2 automatically use exact lightweight geometry for analytic faces.

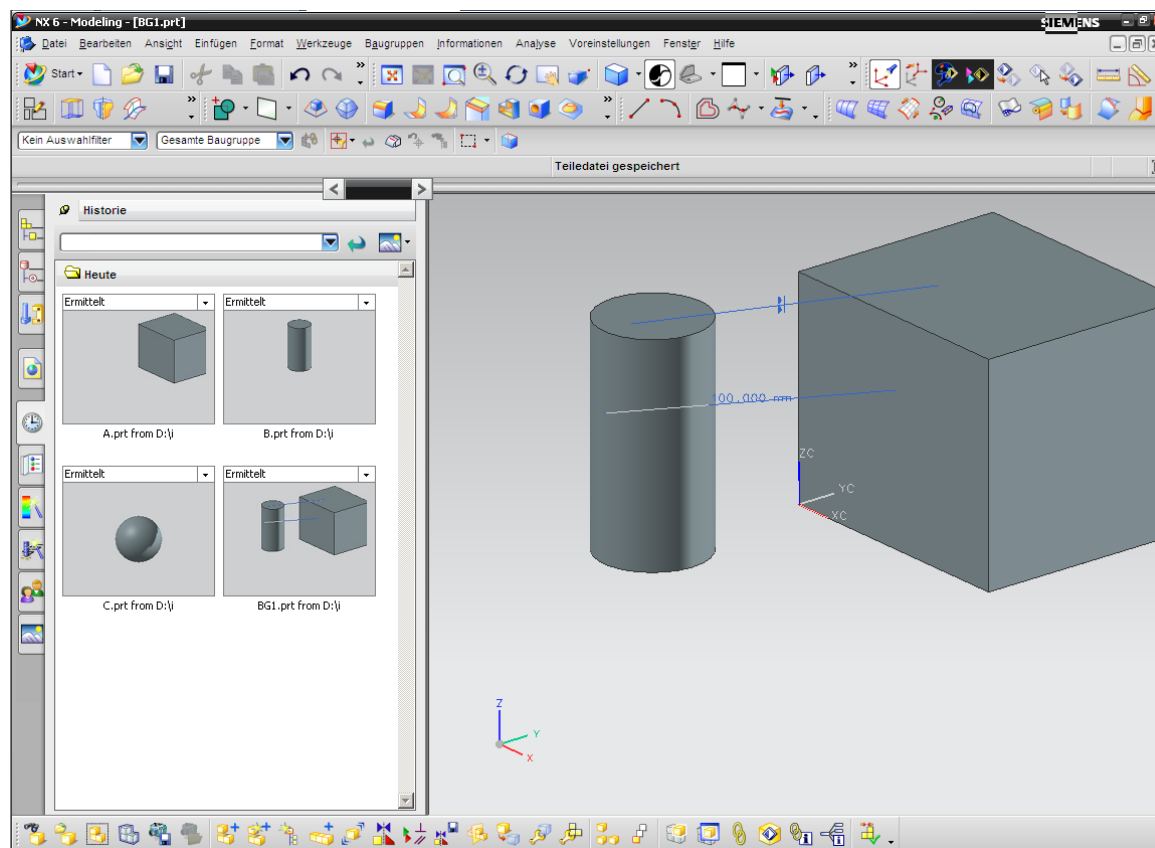
To update other parts to take advantage of the enhanced lightweight representation format, you can run the refile_part utility (in native NX) or the ugmanager_refile utility (in Teamcenter Integration), using the new switches, from the command line of your operating system.

See the *Utilities and File Management Help* and the *Teamcenter Integration for NX Help* for more

NX Load Options

■ allow_substitution Native

Ausgangssituation:



Ermöglicht, dass die Baugruppe mit einer Komponente mit einem falschen internen Identifikator (aber dem korrekten Namen) geladen wird, obwohl es sich um ein völlig anderes Teil handelt.

Hinweis:

Besitzen die neue Komponente und die ersetzte Komponente keine gemeinsame Historie, so enthält auch die Aktualisierung keinerlei gemeinsame Historie, so dass alle mit der ursprünglichen Komponente verknüpften Daten verloren gehen.

Nach Abschluss der Öffnungsoperation gibt NX einen Bericht aus, aus dem hervorgeht, dass die Ersetzung stattgefunden hat.

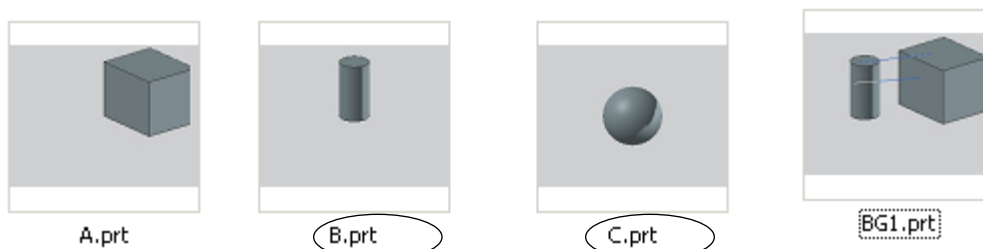
Allow Replacement (Ersatz zulassen)



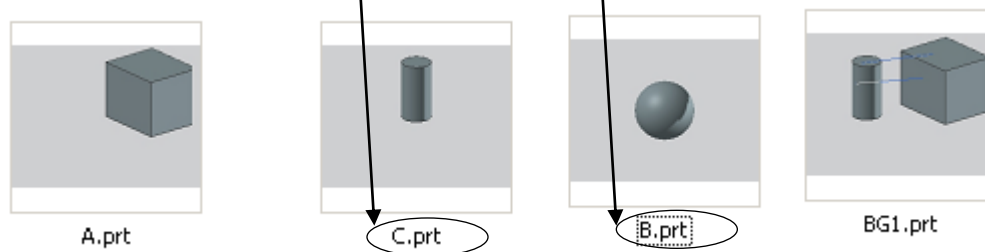
NX Load Options

■ allow_substitution

Ausgangssituation:



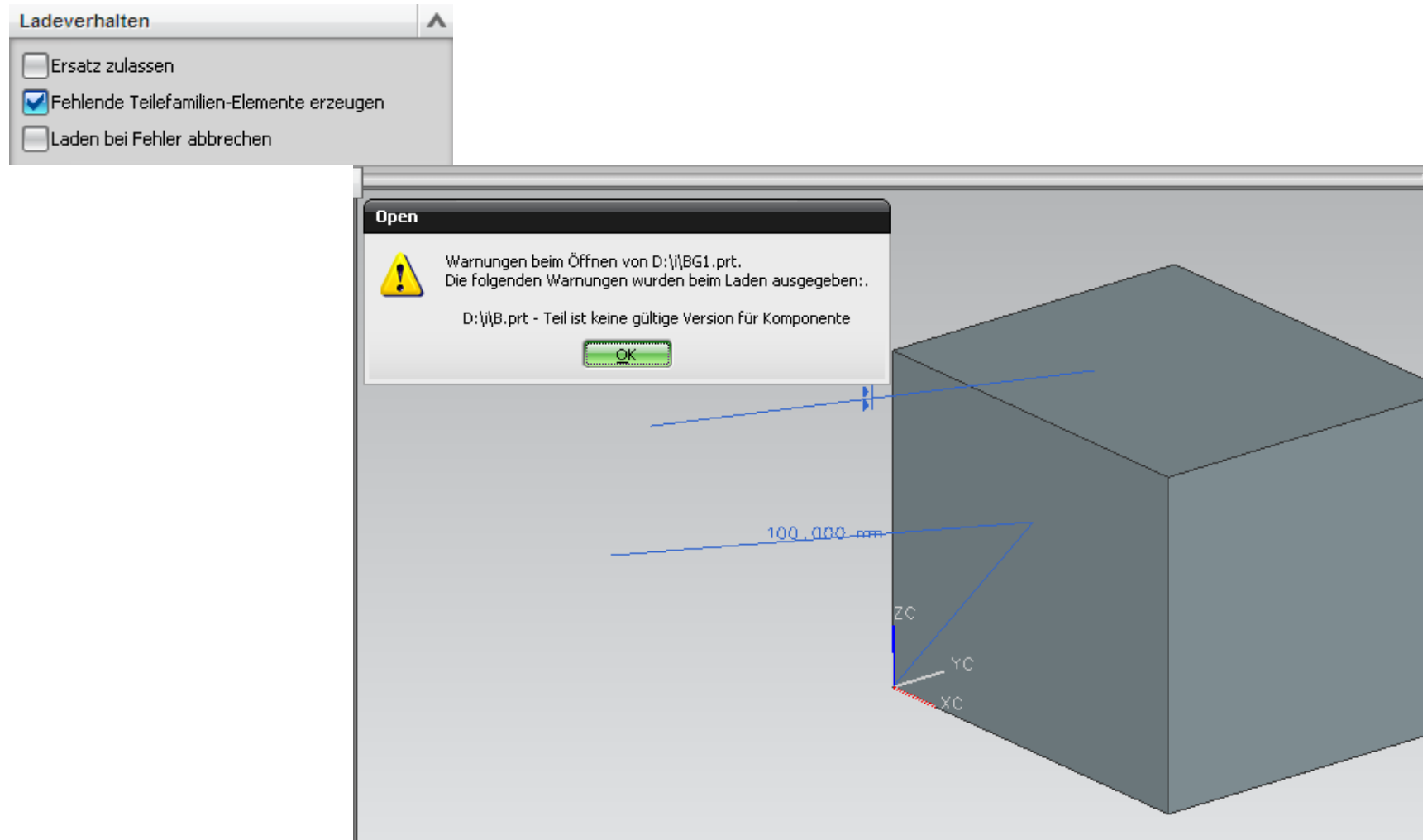
Umbenennung von
C.prt nach B.prt und B.prt nach C.prt





NX Load Options

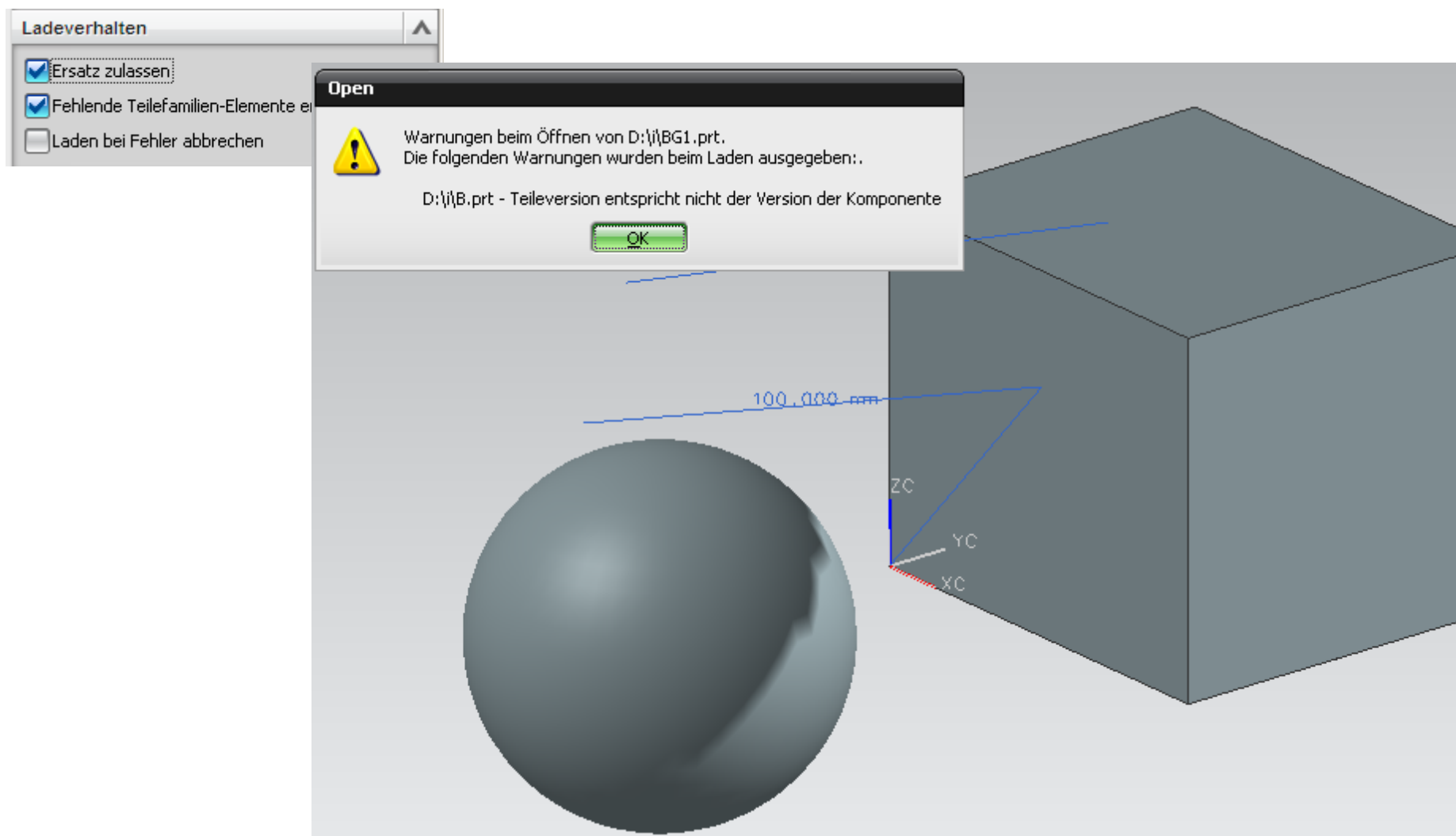
- Laden der BG1.prt mit allow_substitution=no





NX Load Options

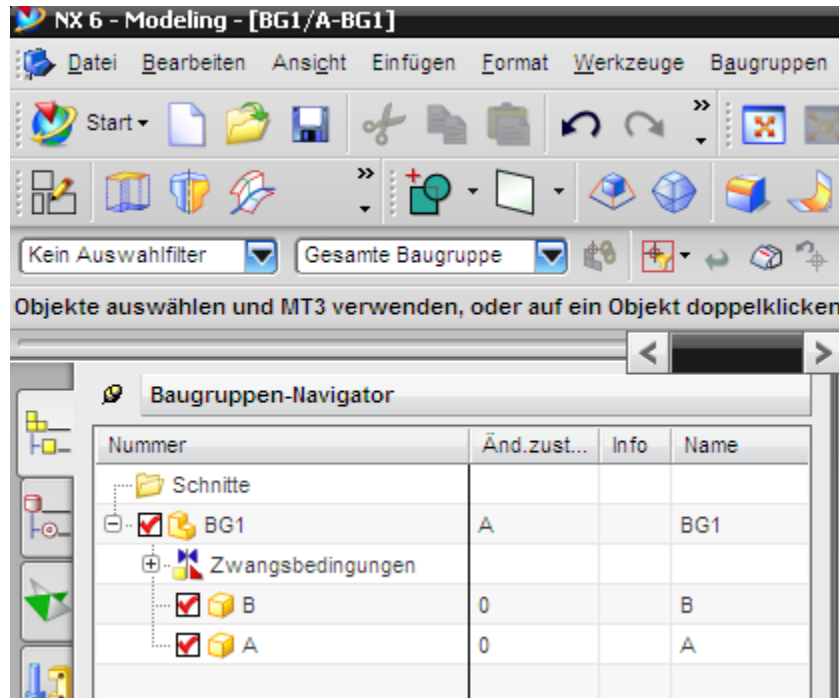
- Laden der BG1.prt mit allow_substitution=yes



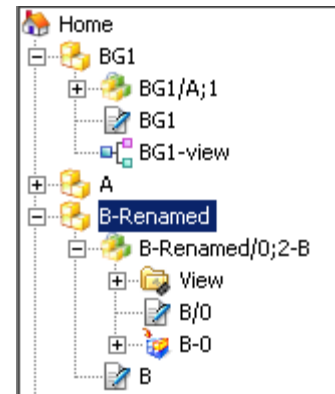
NX Load Options

- allow_substitution Tce Überprüfung nach dem umbenennen der ItemID

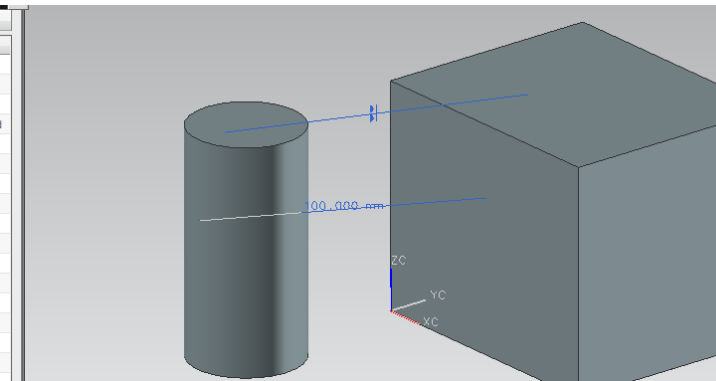
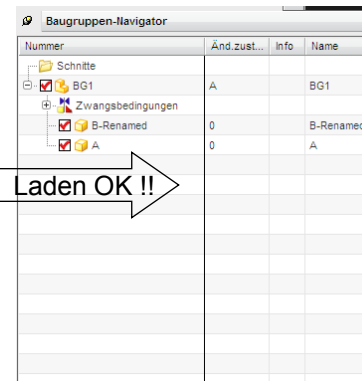
Ausgangssituation:



Situation nach umbenennen der Item ID



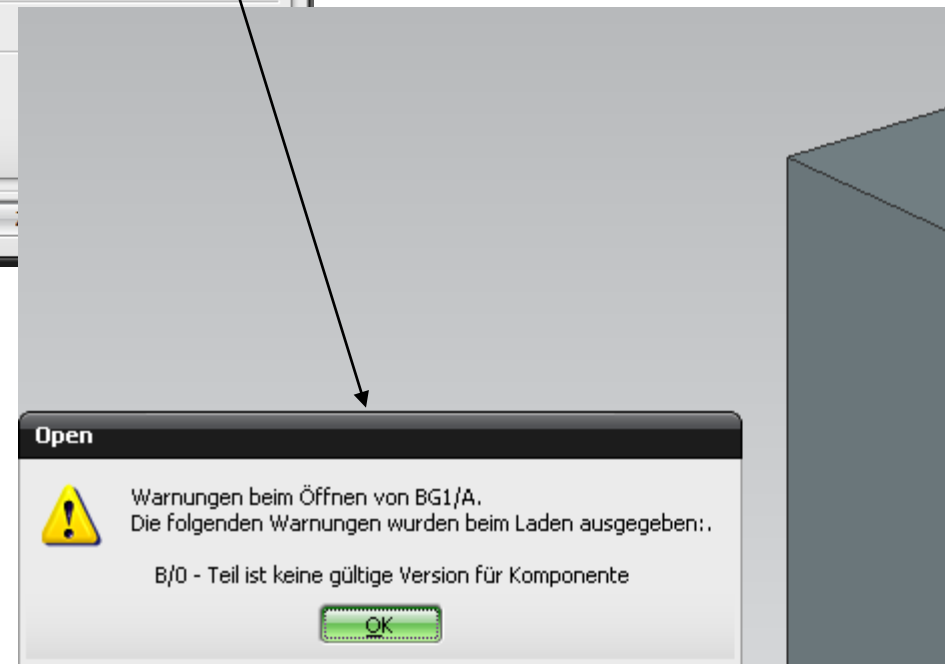
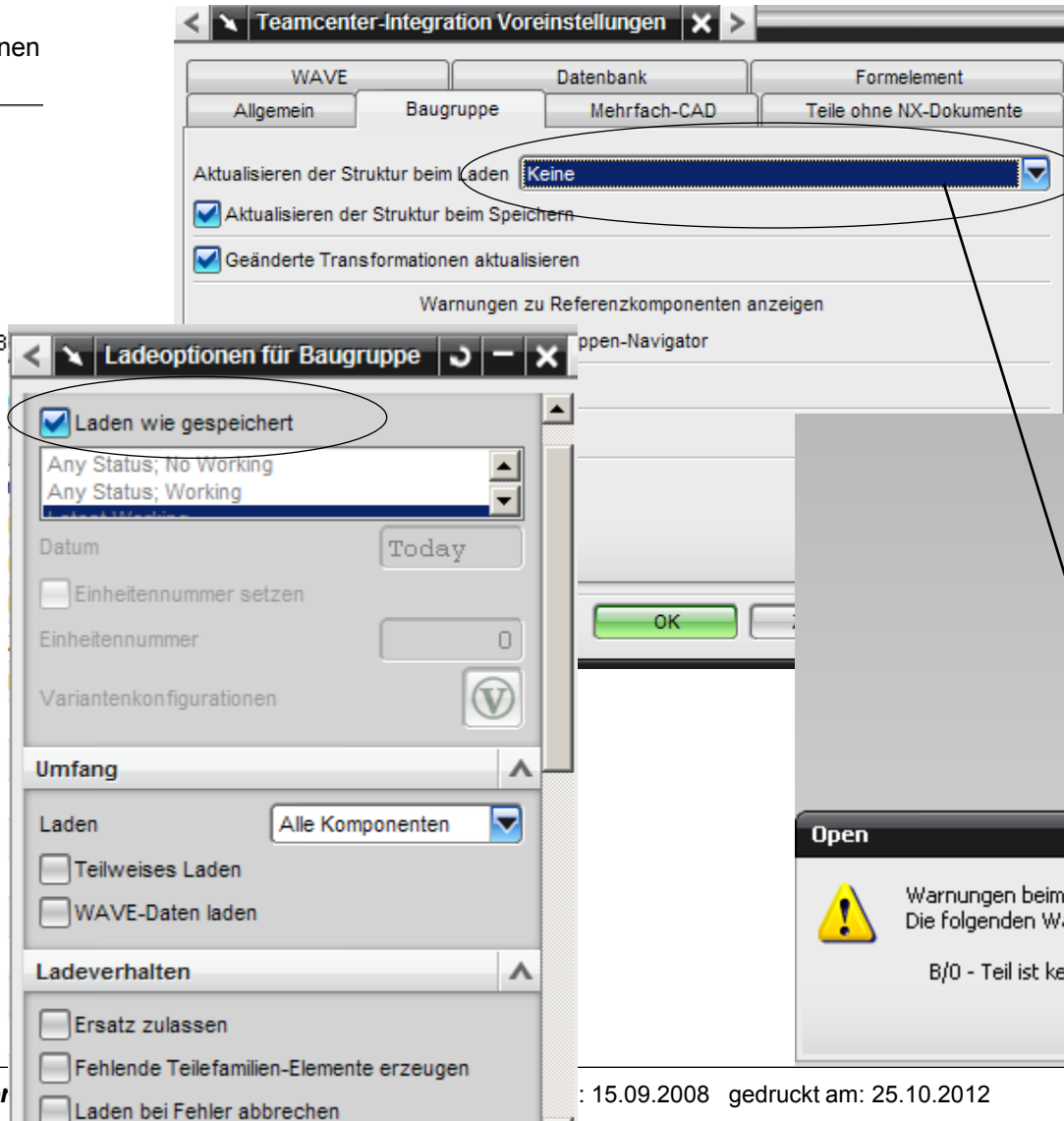
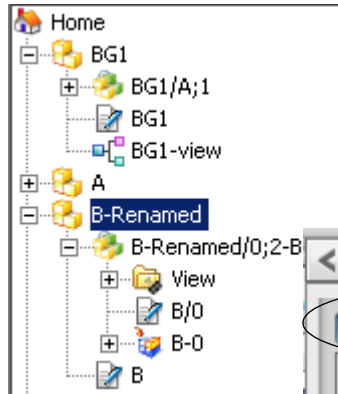
Laden OK !!



NX Load Options

- allow_substitution Tce Überprüfung nach dem umbenennen der ItemID

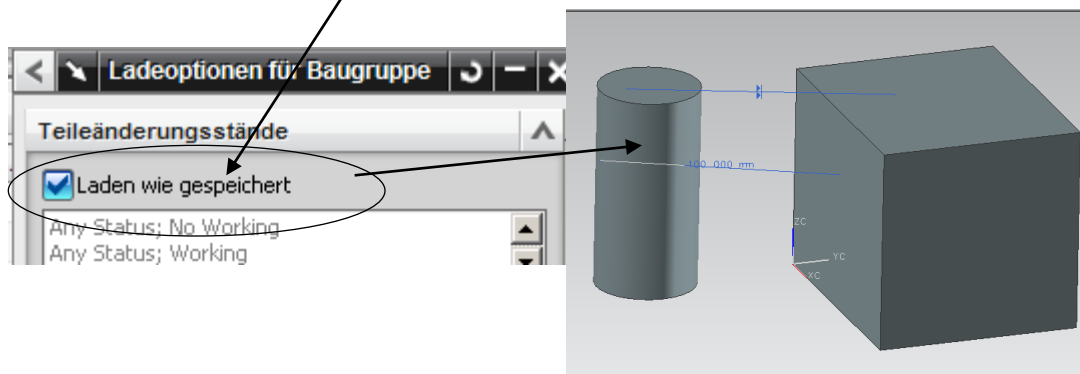
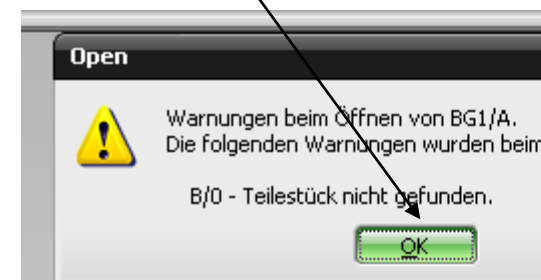
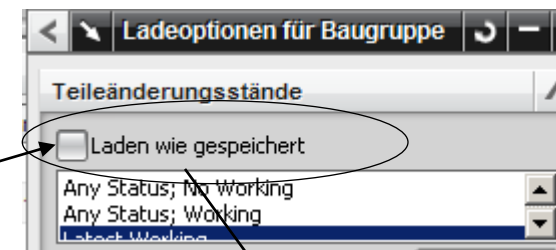
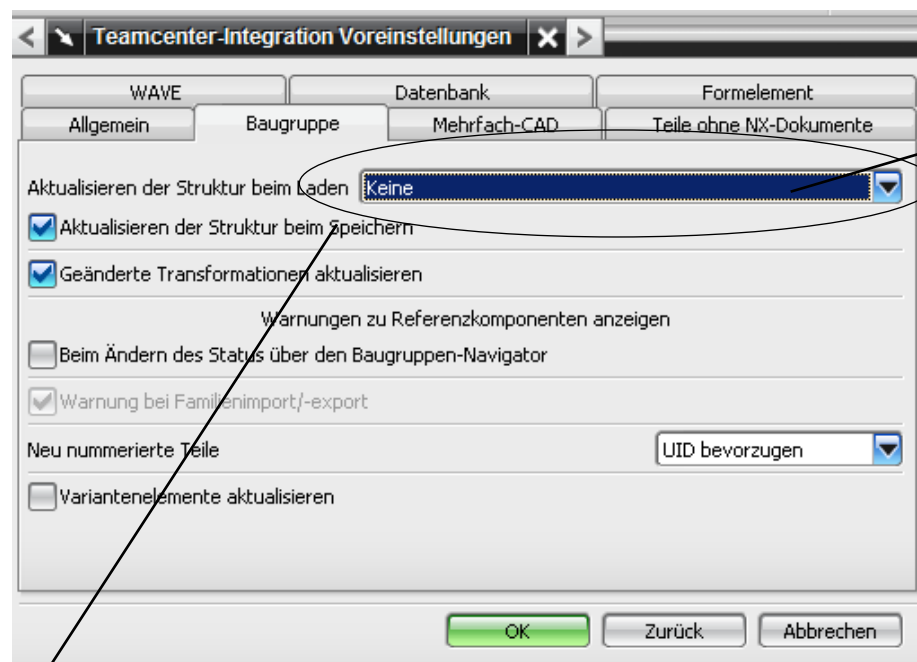
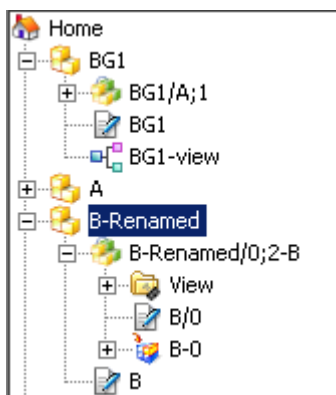
Situation nach umbenennen der Item ID



NX Load Options

- allow_substitution Tce Überprüfung nach dem umbenennen der ItemID

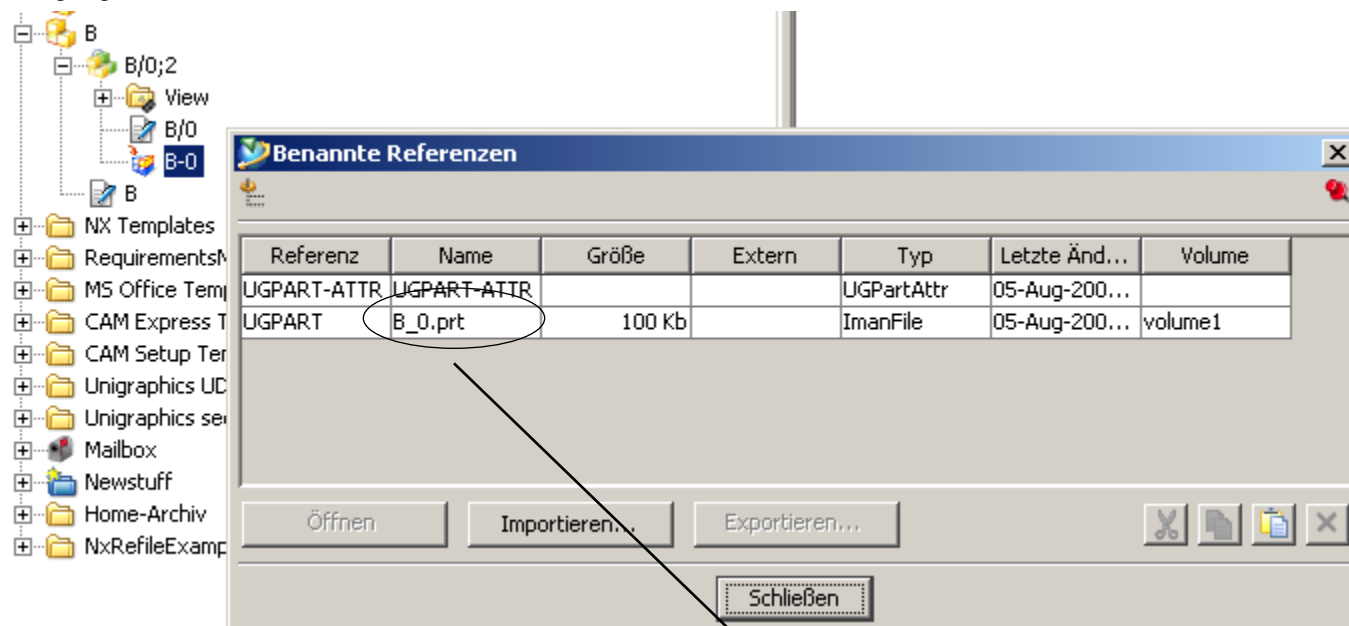
Situation nach umbenennen der Item ID



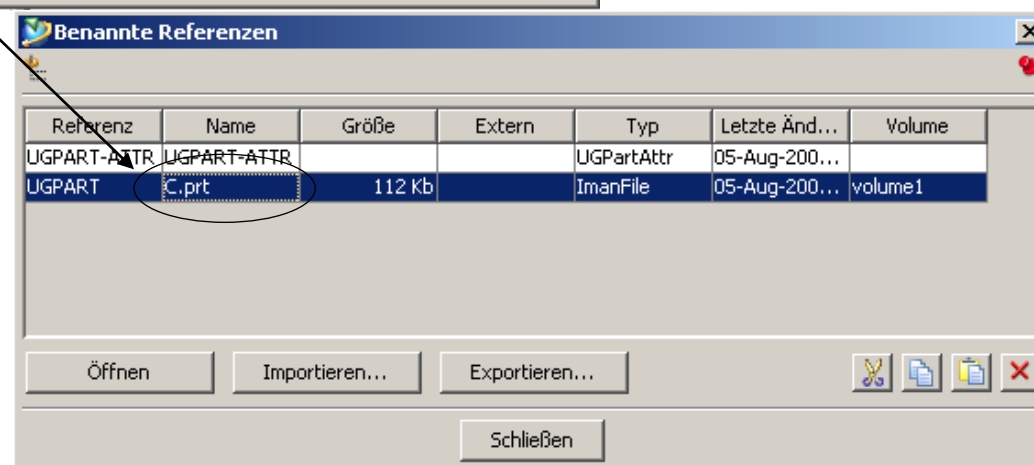
NX Load Options

- allow_substitution Tce Überprüfung nach dem Austausch von B durch anderes Part via Benannte Referenz

Ausgangssituation:



Situation nach austauschen
von B_0.prt durch C.prt





NX Load Options

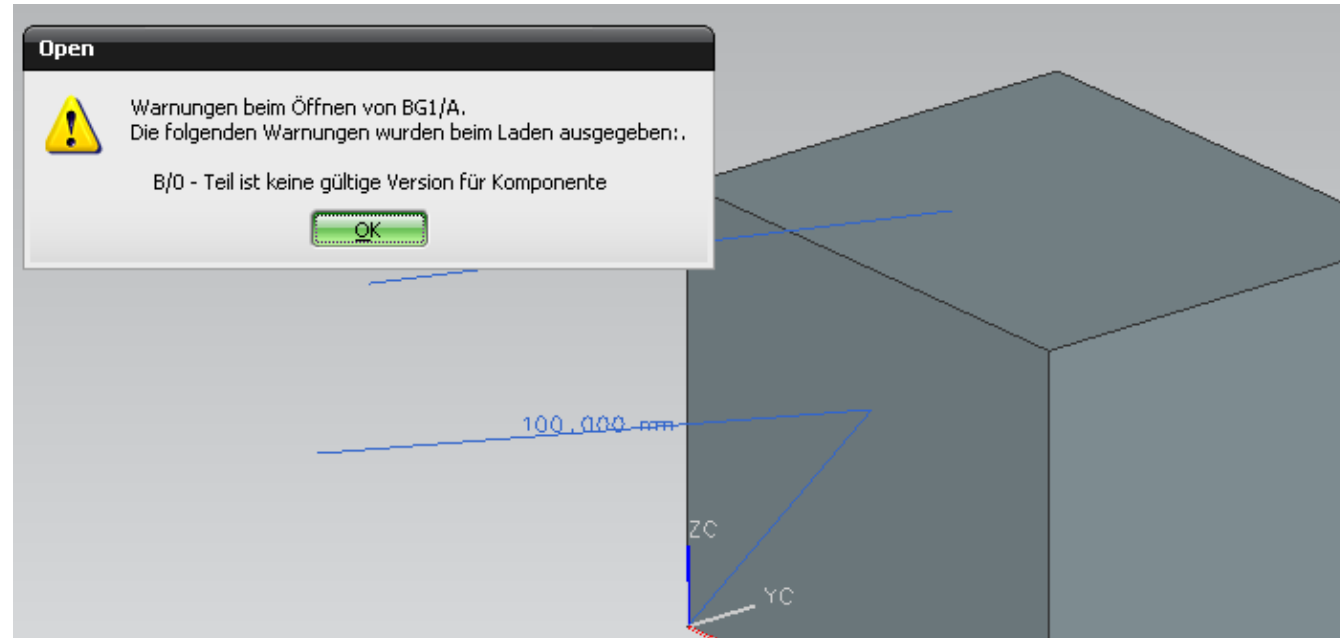
- allow_substitution Tce Überprüfung nach dem Austausch von B durch anderes Part via Benannte Referenz

Laden mit Einstellung:

- ☐ Ersatz zulassen
☐ Fehlende Teilefamilien-Elemente erzeugen
☐ Laden bei Fehler abbrechen

Ergebnis:

<div>  Schnitte </div>	
<div>  BG1 </div>	A
<div>  Zwangsbedingungen </div>	
<div>  B </div>	0
<div>  A </div>	0

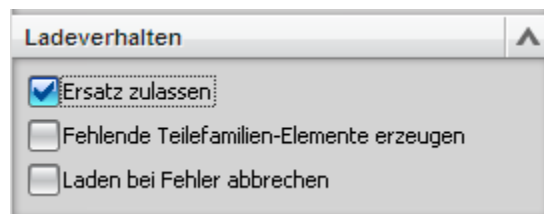




NX Load Options

- allow_substitution Tce Überprüfung nach dem Austausch von B durch anderes Part via Benannte Referenz

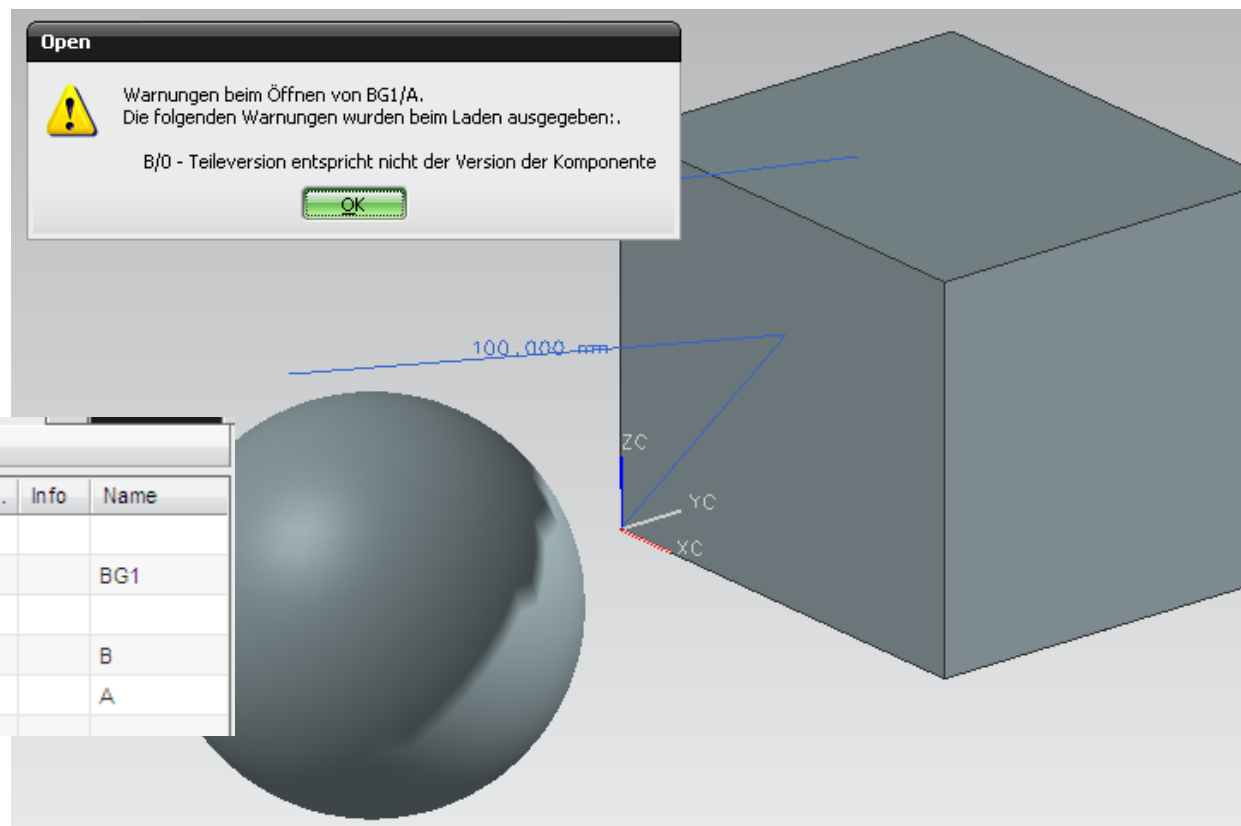
Laden mit Einstellung:



Ergebnis:

Baugruppen-Navigator

Nummer	Änd.zust...	Info	Name
[-] Schnitte			
[+] BG1	A		BG1
Zwangsbedingungen			
B	0		B
A	0		A





NX Load Options

- allow_substitution Tce Überprüfung Refile nach dem Austausch von B durch anderes Part via Benannte Referenz

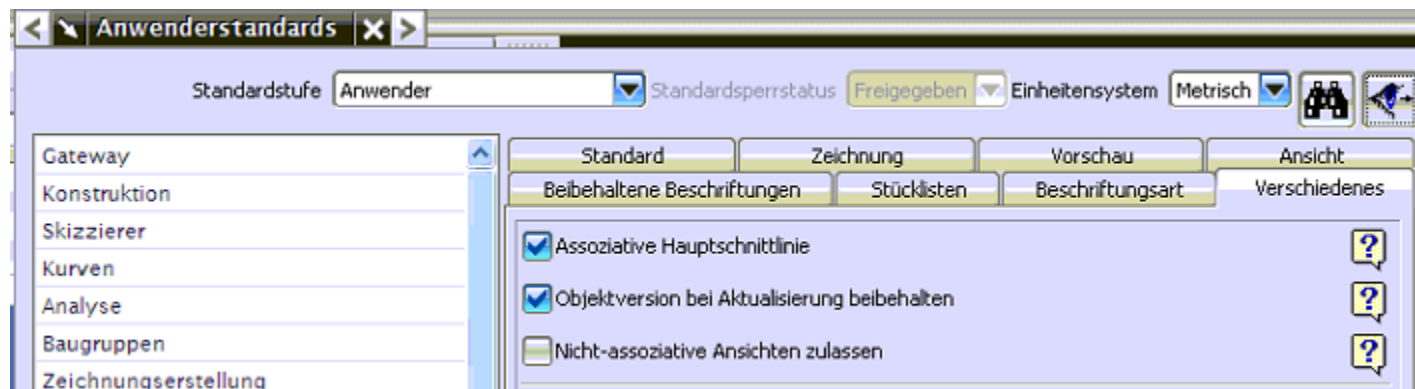
LoadOptions_LoadSubstitution: dont_allow_substitution

-keep_volume=yes -update_mod_props=no -non_masters=yes -refile_released=yes -bypass=yes -convert_mcs="%temp%_convert_mcs_report.txt" -mc_dry_run



Anwenderstandards Objektversion bei Aktualisierung beibehalten

■ Eingetragen 29.09.2009



Maintain Object Version On Update

IR Number: 6168456

Angular dimension (2x 20 degrees) in the right view is flipping when opening the part file and updating in NX 6.0.2. The customer was previously instructed to set the "Maintain Object Version On Update" in the Customer Defaults and this is not helping. This dimension continues to flip from a 20 degree angle callout to a 160 degree angle callout. The associativity to the centerline appears correct, and when reassociating is not correcting it.

***** <<<<< This PR has been closed. The resolution was: >>>>> *****

Siemens PLM believes that the problem you reported where an Angular dimension flips when updated will be resolved by code changes made to the upcoming release of our software NX6.0.2. Turning on "Maintain Object versions" in Preference->Drafting->General before updating the view will not give the expected result.

NX6.0.2. Turning on "Maintain Object versions" in Preference->Drafting->General

<Pref Tab="Verschiedenes" name="Drafting_versionObjects" title="Objektversion bei Aktualisierung beibehalten" value="1" Category="Allgemein" modified="2009-09-29T10:00:00">



RF NX7.5

■ Datenzuwachs Eingetragen 12.05.2010

set JobParameter=-d "D:\NxData\W9287151_RevA_Orig" -o "D:\NxData\W9287151_RevA_NX75_Rf" -r -regen_lw

