

User Manual

SolidSteel parametric

for Siemens NX

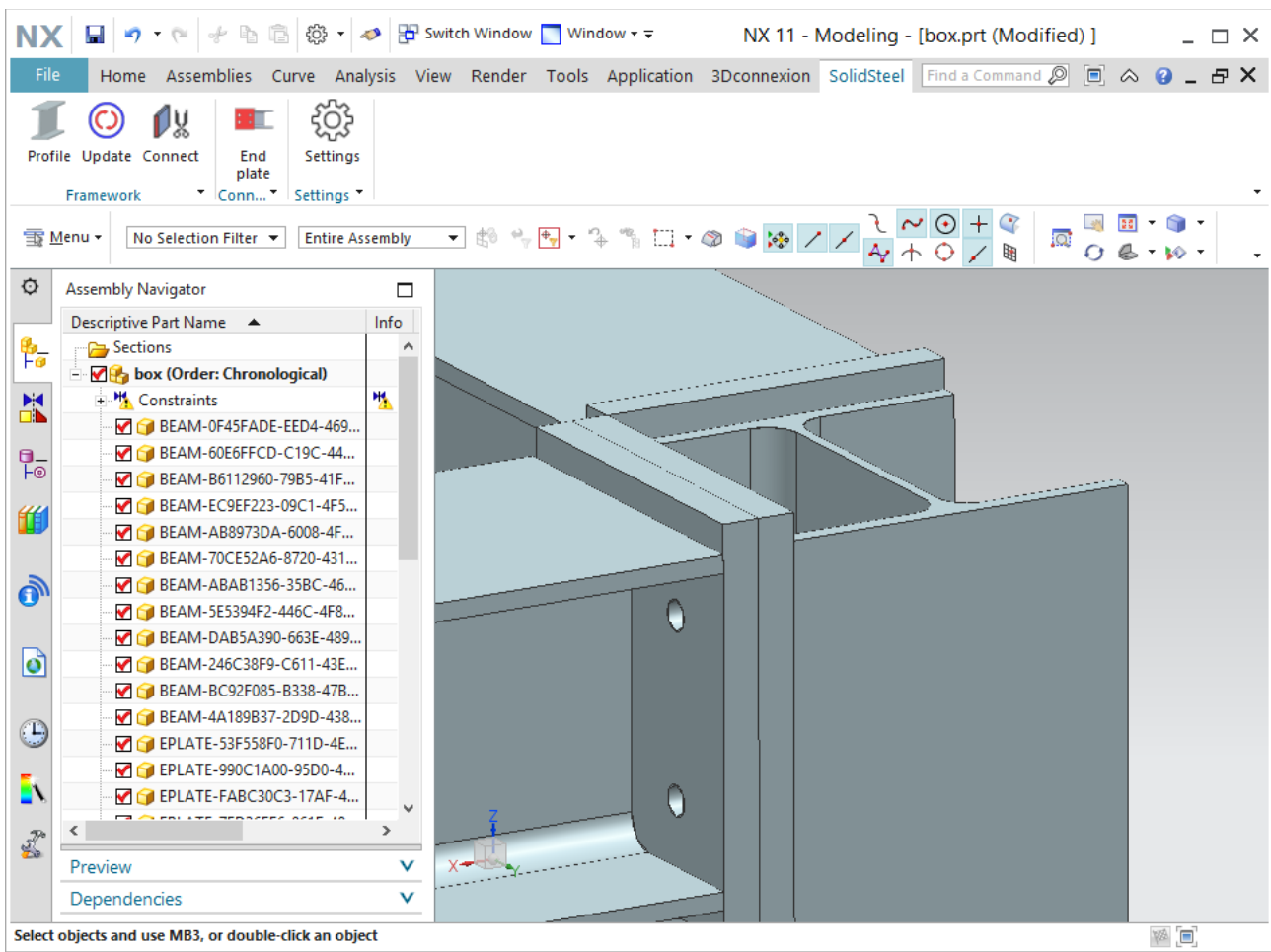


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Introduction

SolidSteel parametric is a 3d steelwork cad system, designed as an add-on module (app) for the standard cad system Siemens NX.

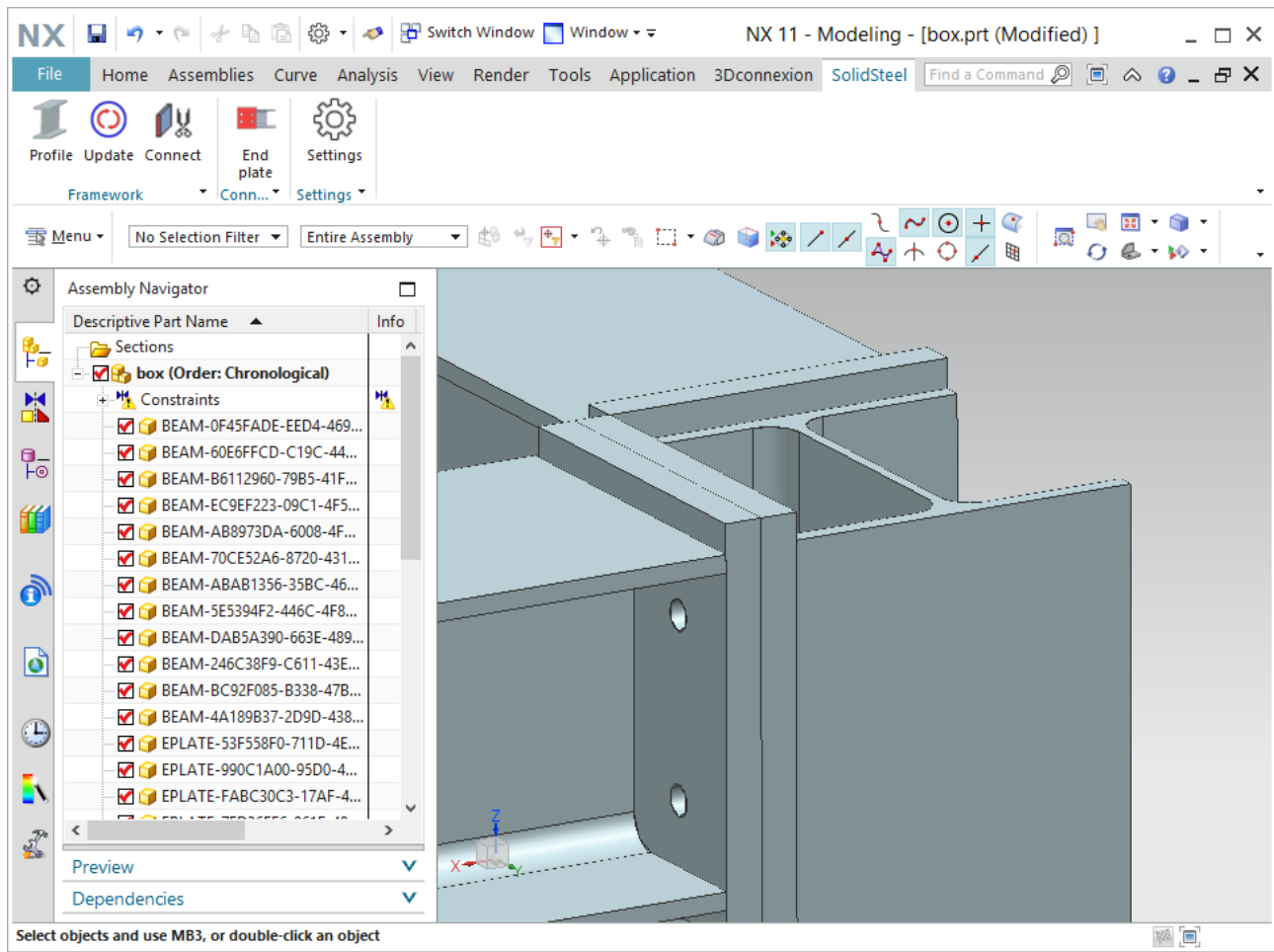
SolidSteel parametric is modular designed and can be used in different extension levels, e.g. Framework, Connect and NC Data.

The following manual describes the usage of SolidSteel parametric for Siemens NX. Knowledges about configuration and usage of the basic system Siemens NX are required.

Siegen (Germany), April 2018

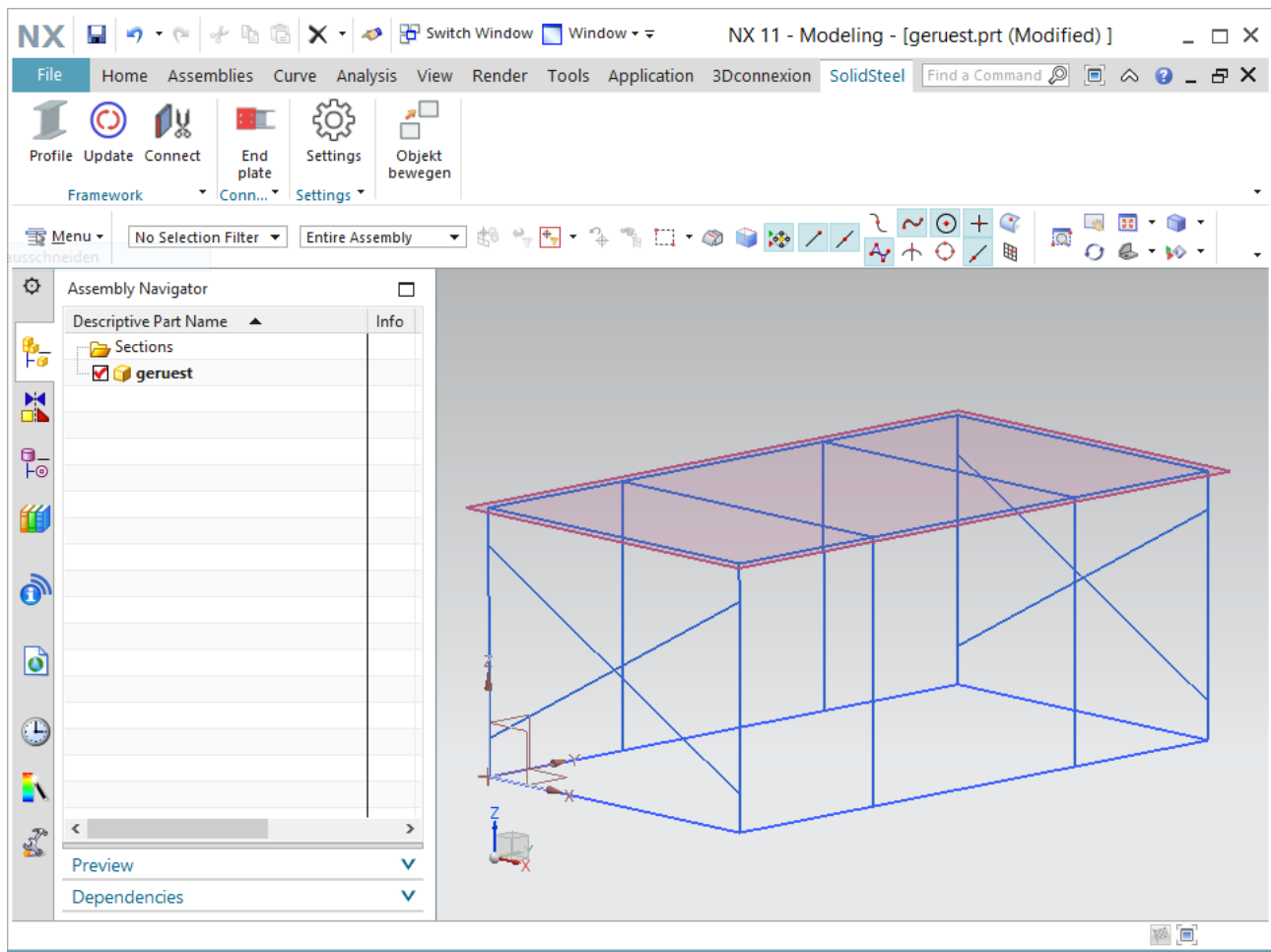
Overview

SolidSteel parametric is embedded into Siemens NX as an Add-On. After the installation you will find SolidSteel parametric in a new Tab called „SolidSteel“ in the Siemens NX ribbon bar.



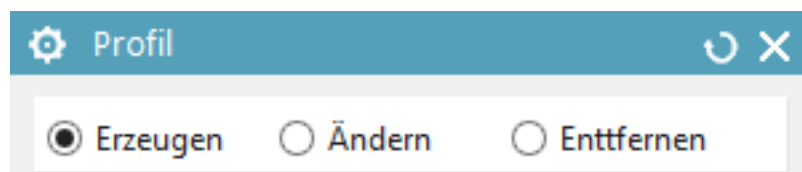
Profiles

Profiles are the basis for a structural steel framework. In SolidSteel parametric, profiles can be placed along existing system lines. So a usual NX sketch can be used.



Usual Siemens NX sketch with systemlines

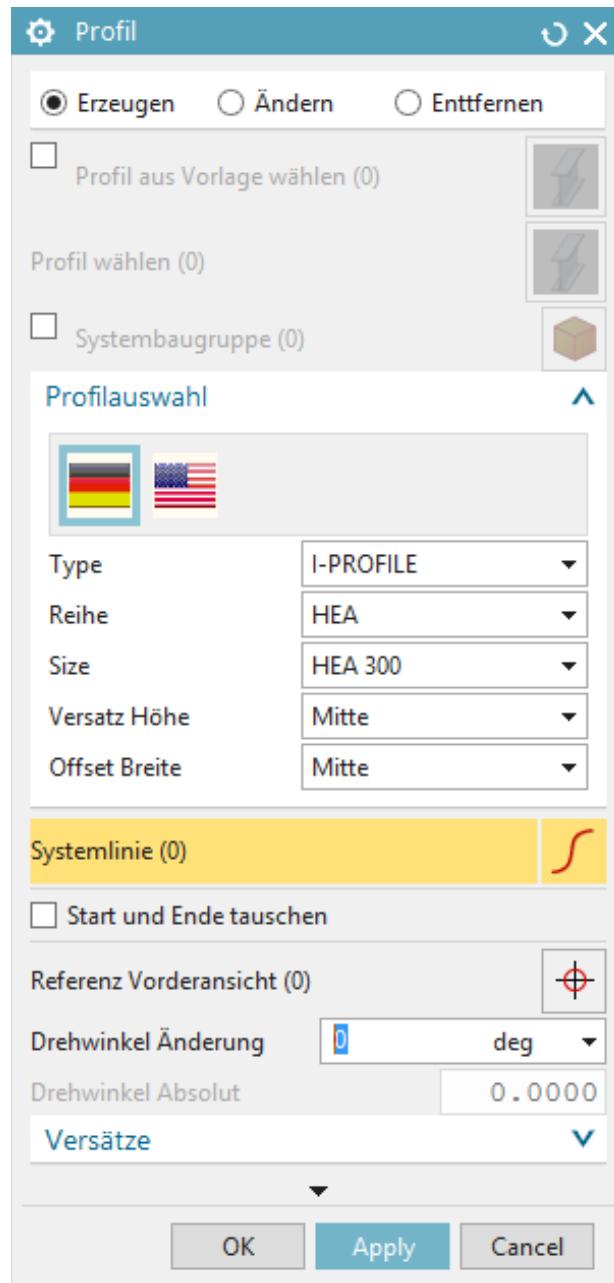
For creating, editing or deleting profiles in SolidSteel parametric, open the profiles dialogue. On top of this dialogue is the top selection menu. Here can be switched between the different dialogue functions.



Top selection menu of SolidSteel parametric

Creating profiles

For creating profiles in SolidSteel parametric, open the profile dialogue and choose „create“ in the top selection menu.



Profile dialogue of SolidSteel parametric

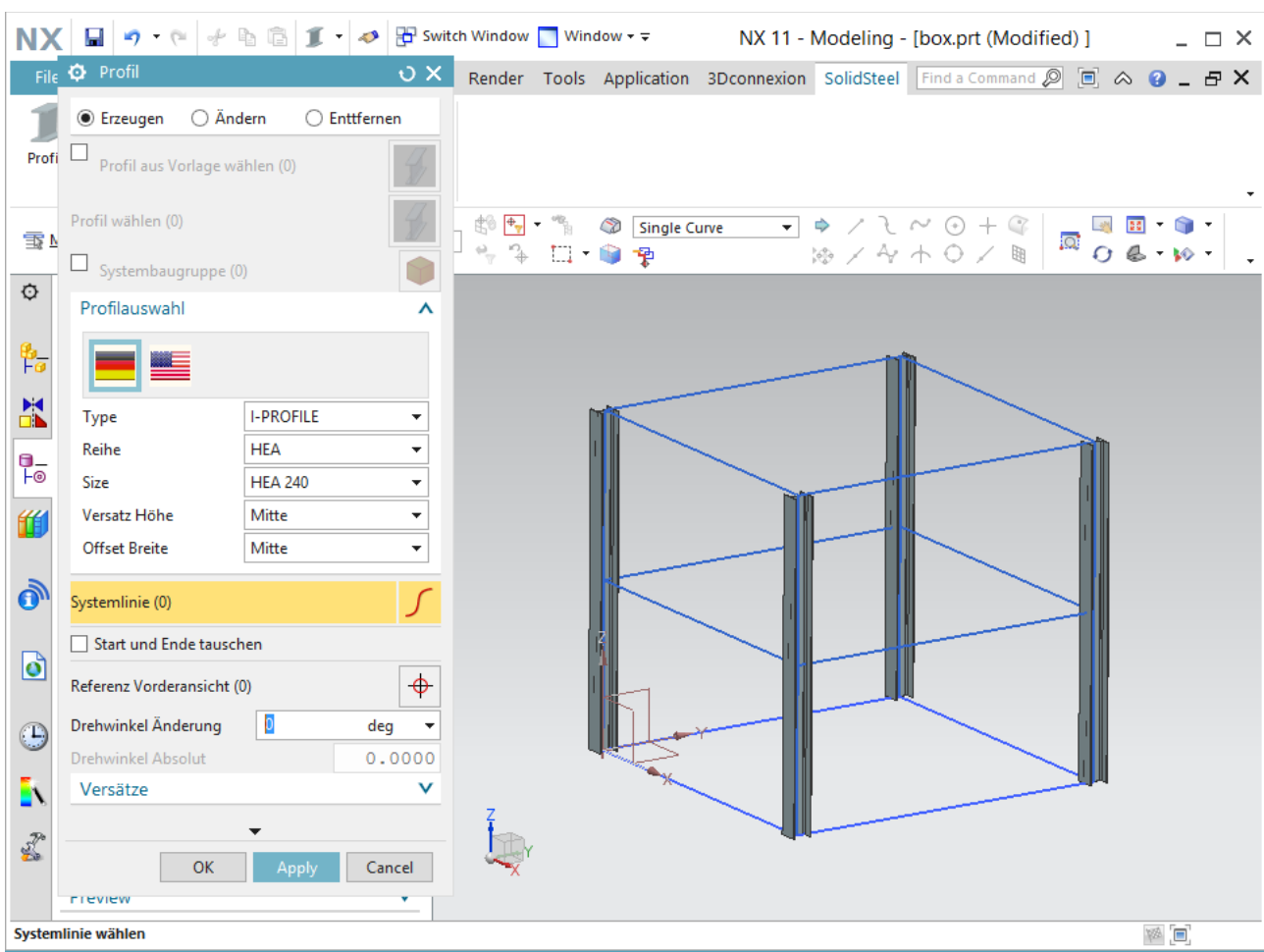
In this menu, there are different options. First you have to choose the location of the profile library. Then choose the profile type, row and size of the profile you want to create.

After that, choose those systemlines, where a profile of this type should be created. Immediately after clicking the system line, a preview of the profile appears.



Selection of 4 systemlines for creation of profiles

Now all the settings can still be changed or modified. Even the profile type or the profile size still can be changed. After clicking apply, the profiles will be created.



4 profiles created on a Siemens NX sketch

Creating profiles from a template

There are many settings you can do before creating a profile. In SolidSteel parametric we can pick those settings from other existing profiles. So we can save time during the construction.

To do that, click the check box „profile from template“ on top of the profile dialogue. Then click the profile in the construction which settings should be copied.

Profil

☒ Erzeugen ☐ Ändern ☐ Enttfernen

☒ Profil aus Vorlage wählen (0)

Profil wählen (0)

☐ Systembaugruppe (0)

Profilauswahl

Type: I-PROFILE

Reihe: HEA

Size: HEA 360

Versatz Höhe: - Höhe/2

Offset Breite: Mitte

Systemlinie (0)

☐ Start und Ende tauschen

Referenz Vorderansicht (0)

Drehwinkel Änderung: 0 deg

Drehwinkel Absolut: 0.0000

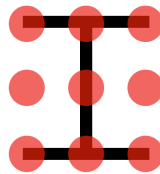
Versätze

OK Apply Cancel

Profile from template function of SolidSteel parametric for Siemens NX

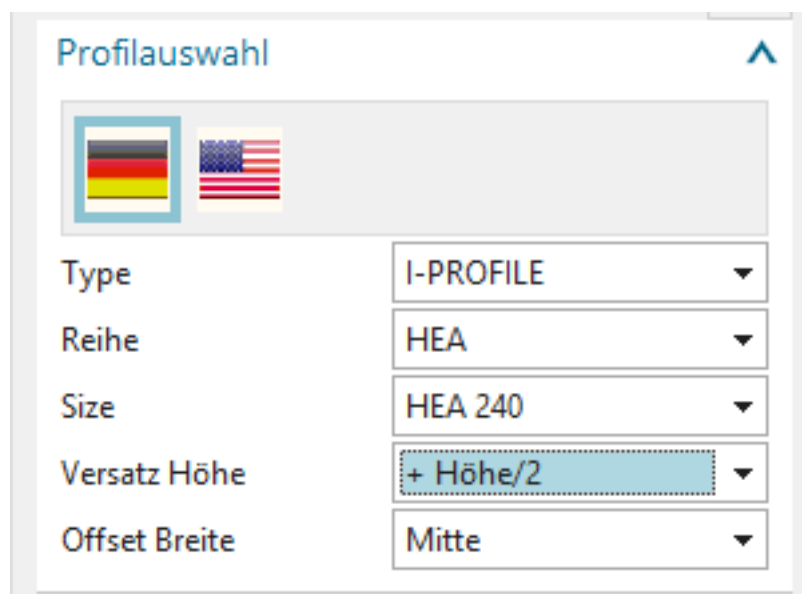
Parallel offsets

In many cases, the system line should not be in the center of the profile. So in SolidSteel parametric there are parallel offsets available. Using this functions, you can place the system line in 9 different points of the profile.

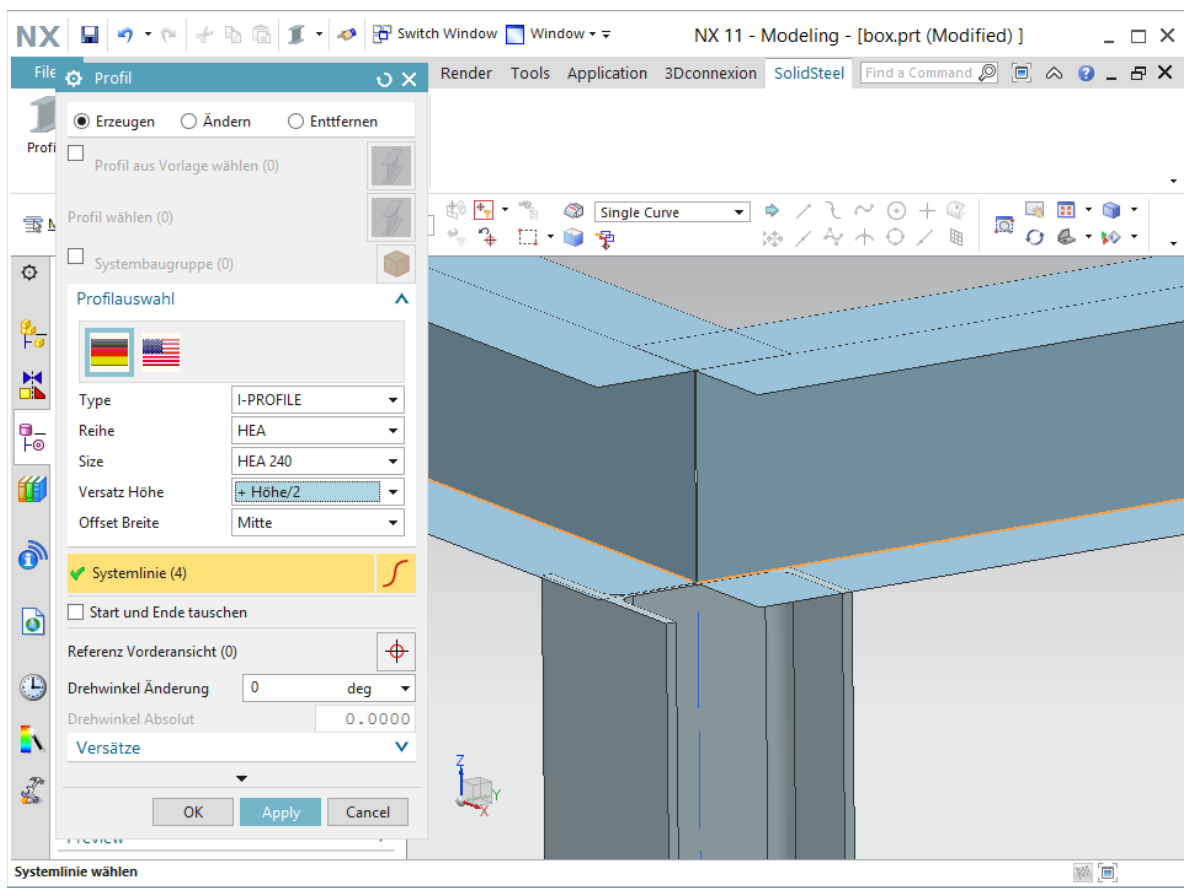
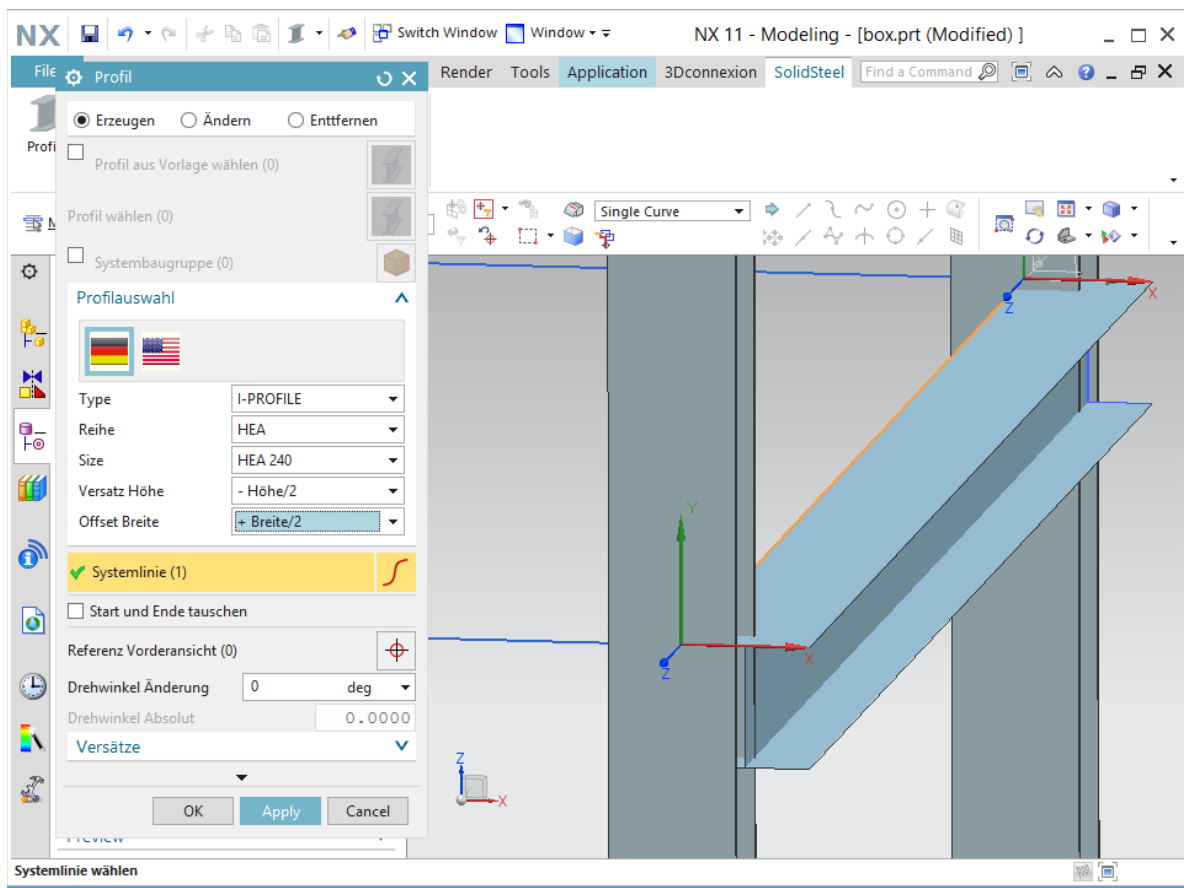


possible parallel offsets in SolidSteel parametric

You can choose the offsets in the profile dialogue.



Setting parallel offsets for a HEA 240



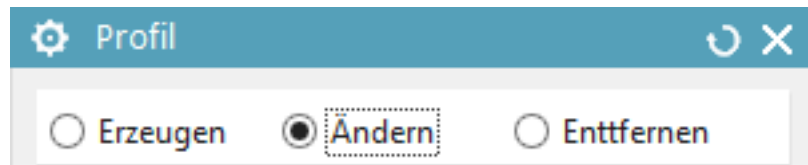
Individual offsets

Not available today

The individual offsets of profiles are not available in this software version. The functions will be available within one of the next upcoming updates.

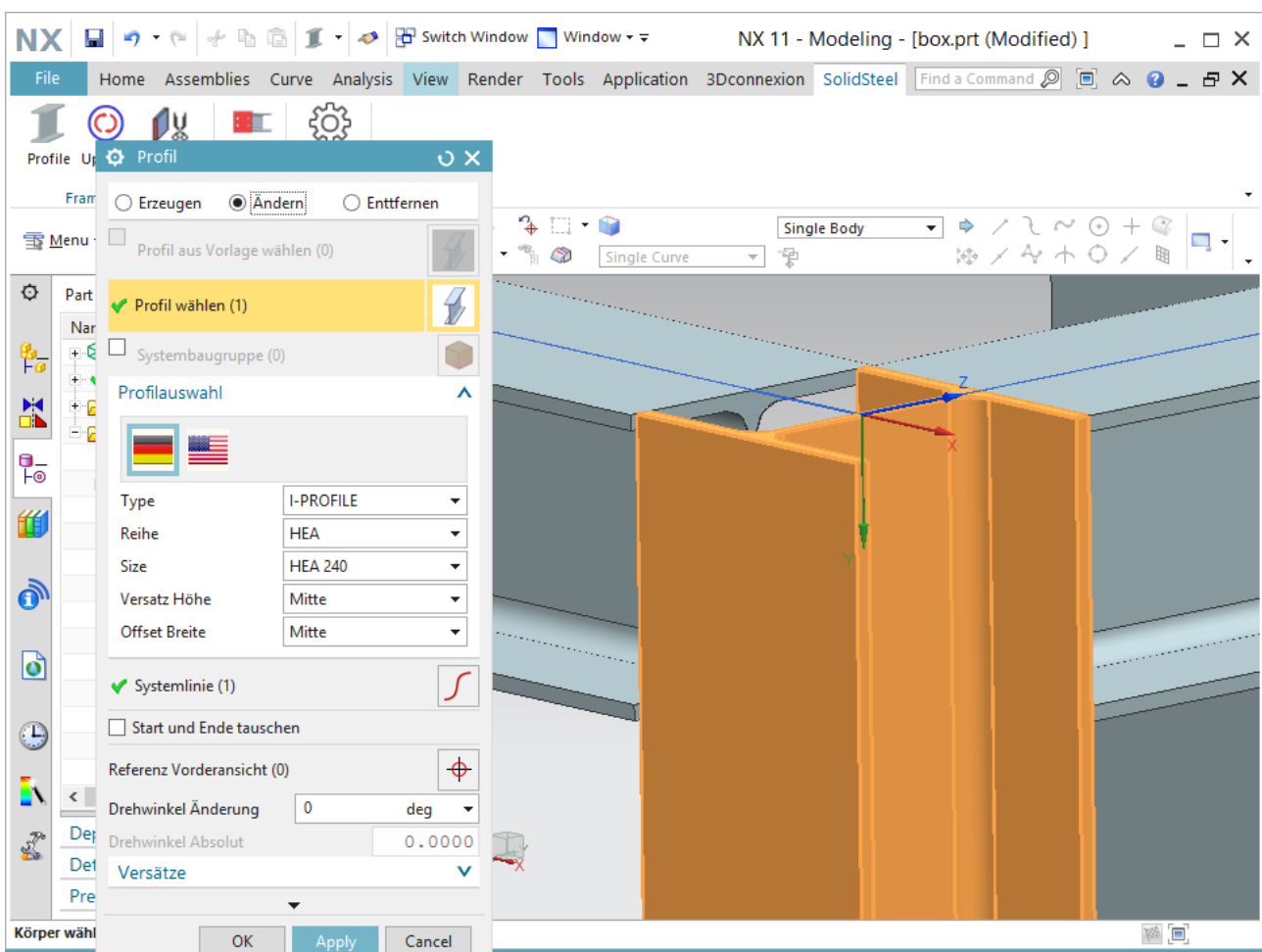
Edit profiles

For editing profiles in SolidSteel parametric, open the profile dialogue and choose „edit“ in the top selection menu.



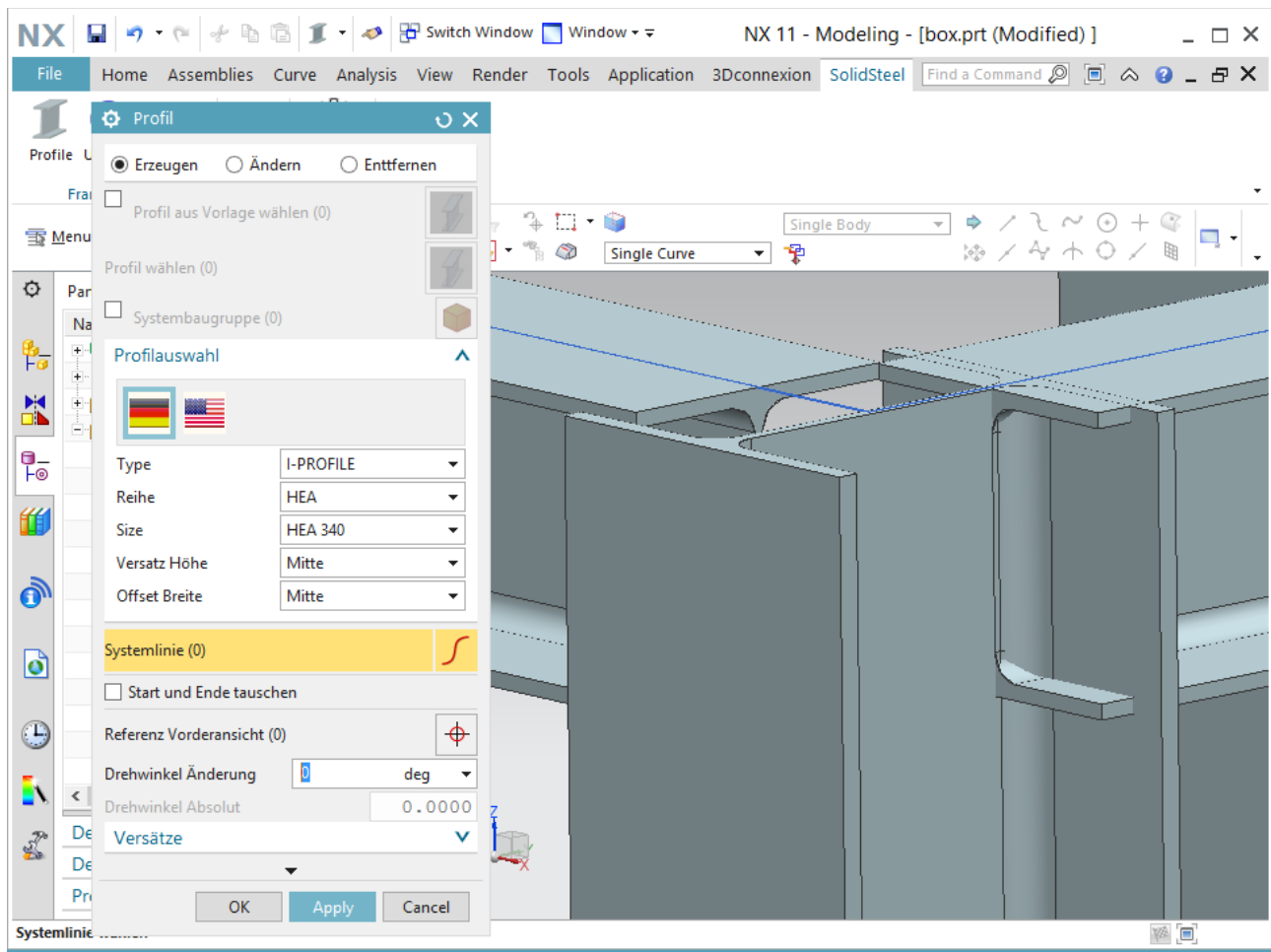
Edit profiles in the top selection menu

Now choose the profile you want to edit or change.



Selected profile which should be edited or changed

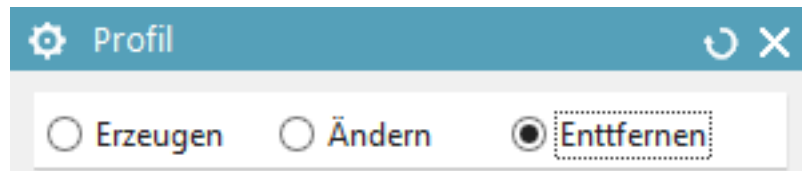
Now all the settings of this profile are in the dialogue and can be changed. After clicking apply, all the changes are set. Maybe an update of the framework is necessary.



A changed profile (Changes: HEA 240 -> HEA 340)

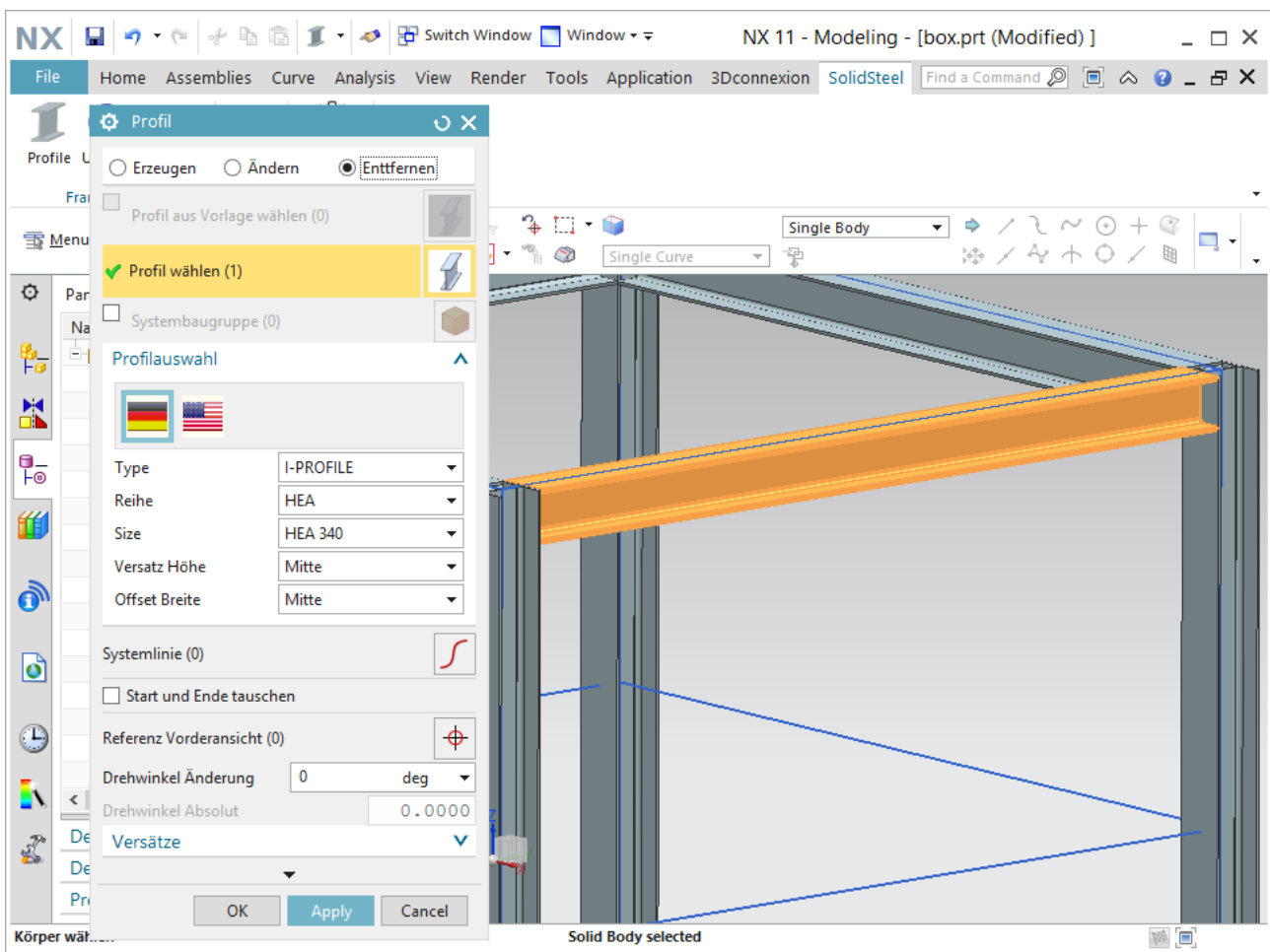
Deleting profiles

For deleting profiles in SolidSteel parametric, open the profile dialogue and choose „delete“ in the top selection menu.

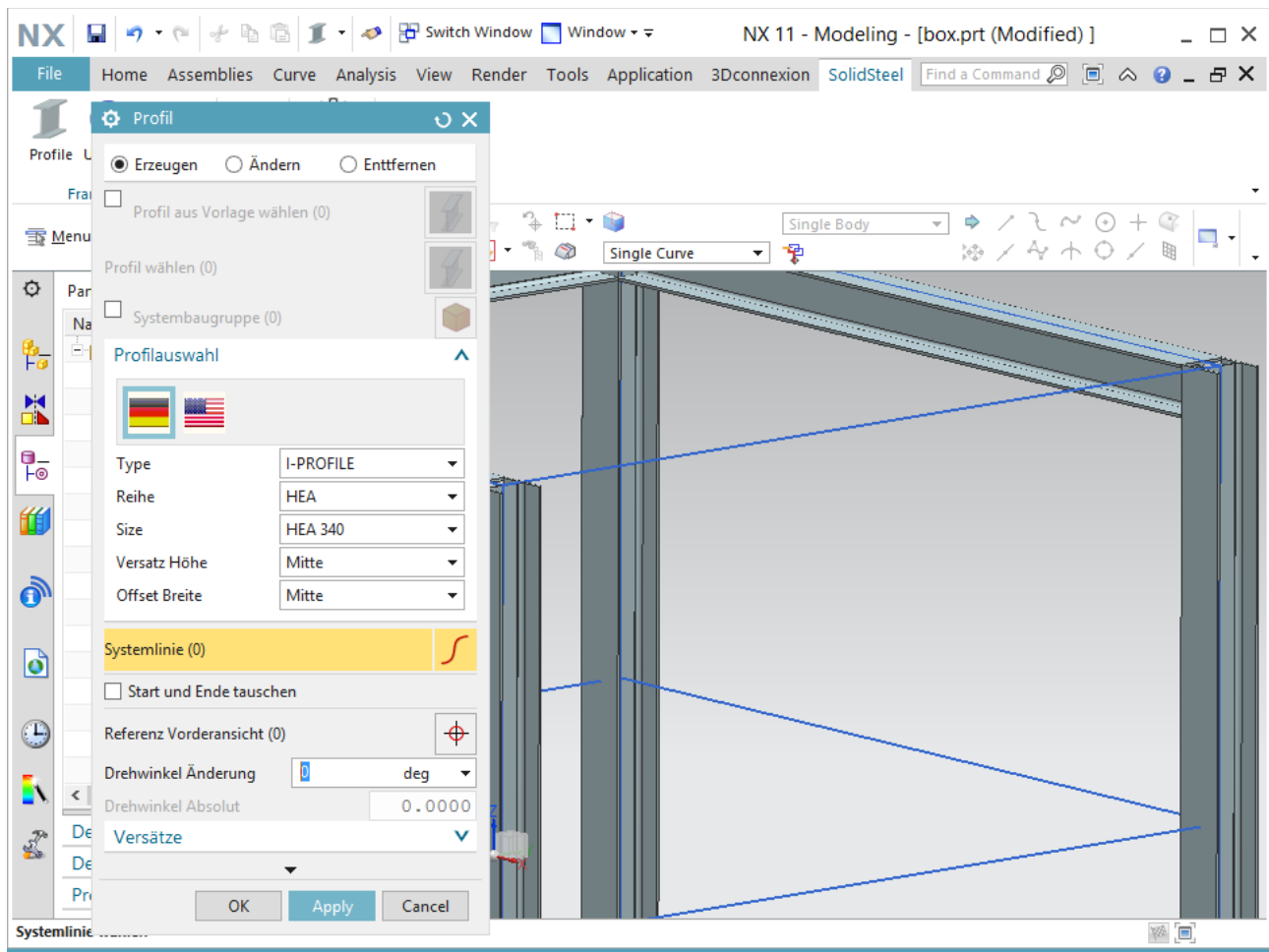


Delete profiles in the top selection menu

For deleting a profile, just select the profile and click apply. Now the profile is deleted. Maybe an update of the framework is necessary.



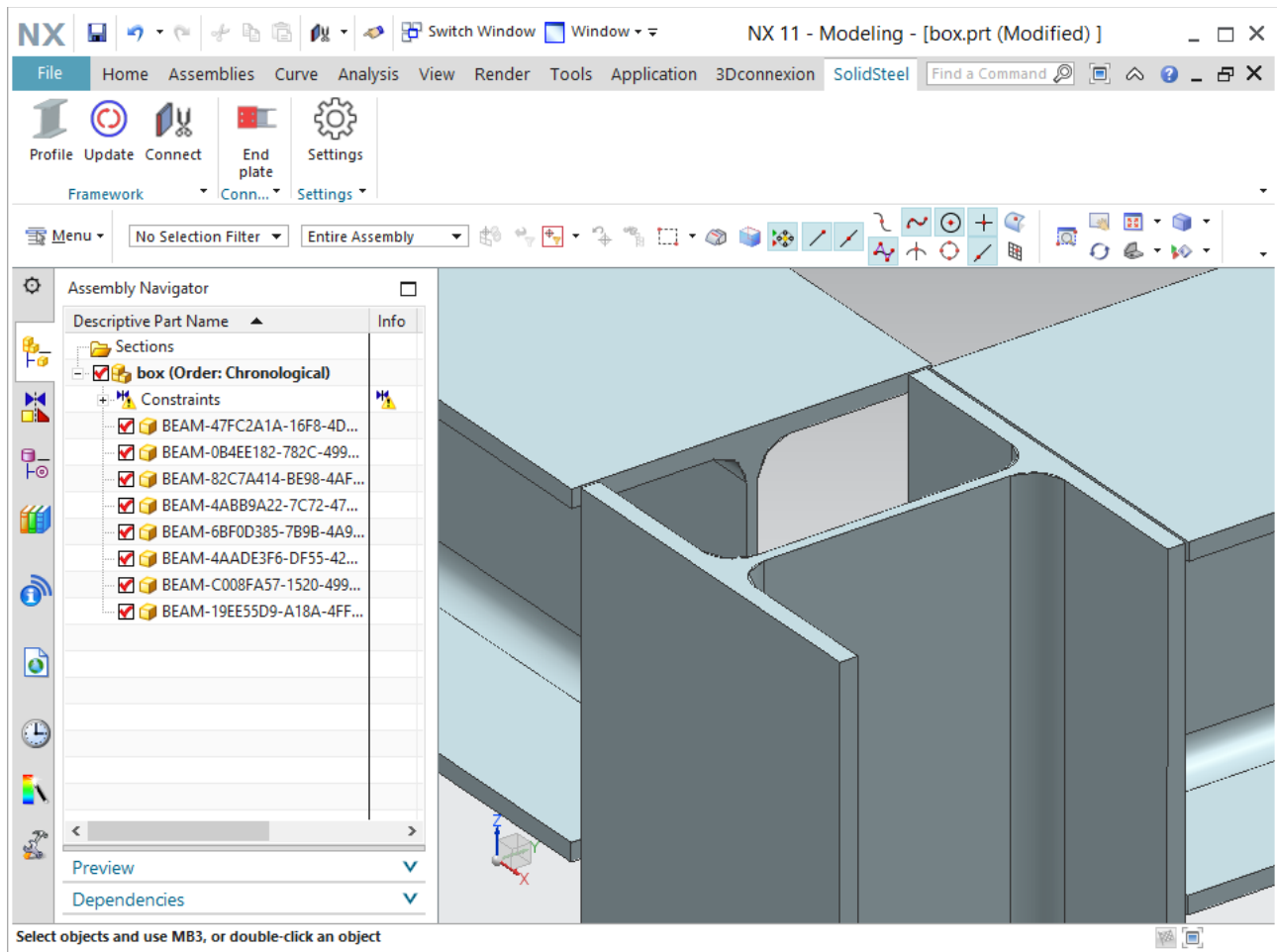
Selected profile should be deleted



Profile is deleted

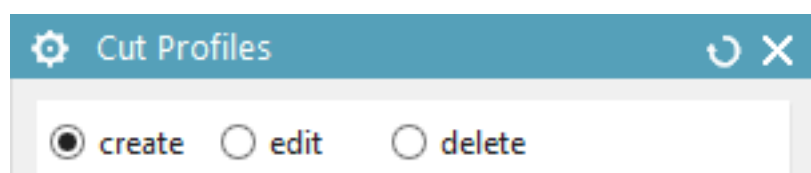
Cutting profiles

The simplest connection of profiles is to cut them off. Of course these functions are also available in SolidSteel parametric for Siemens NX.



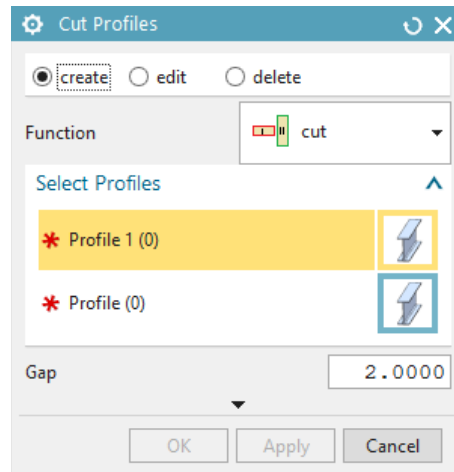
Profiles cut off along each other

To cut profiles, open the „cut profiles“ dialogue and select „create“ in the top selection menu.

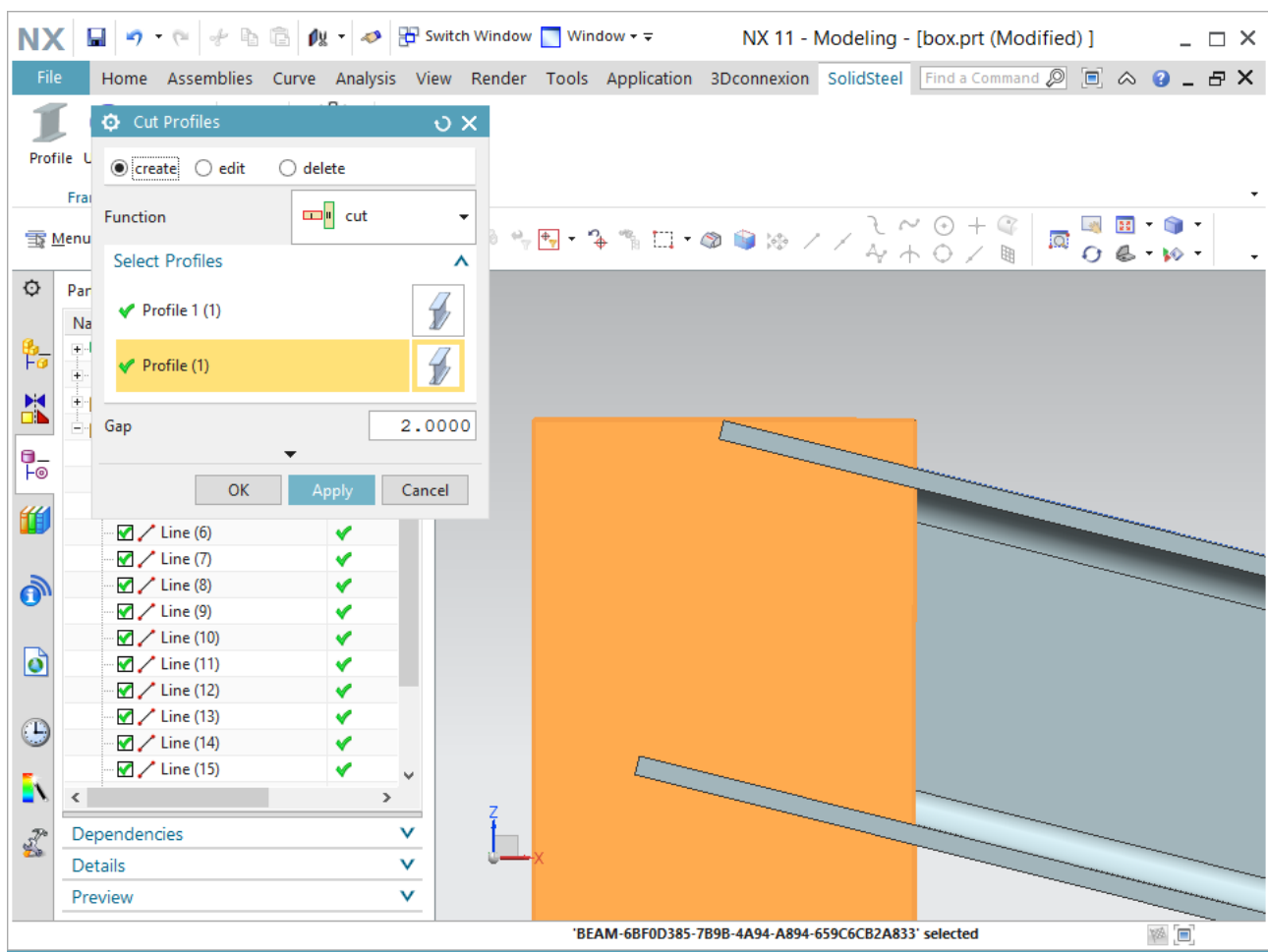


cut

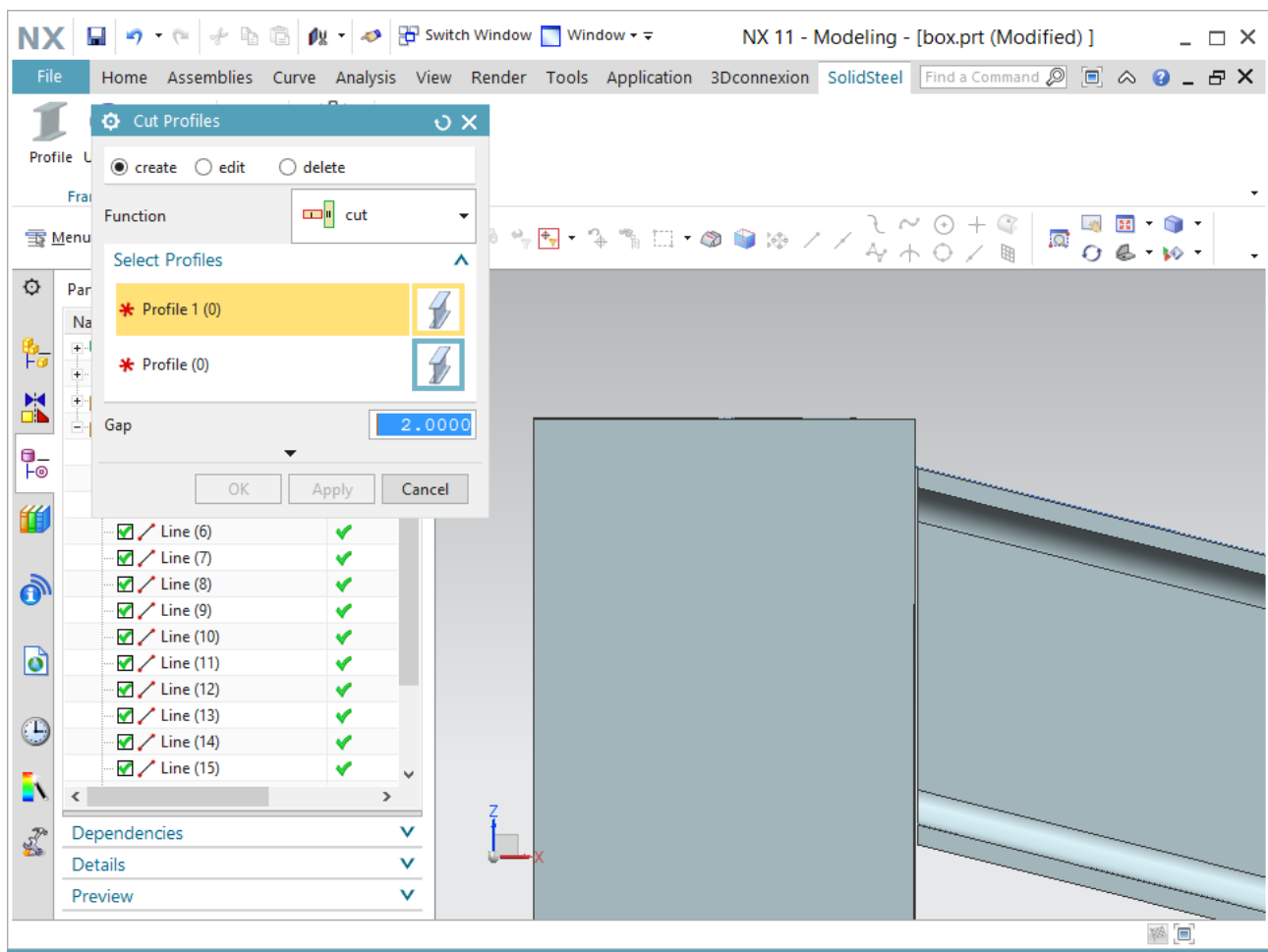
The cut function just cuts the profiles off. To do that select two profiles and click apply. If you need a gap between those two profiles, set the distance.



Cut profiles dialogue



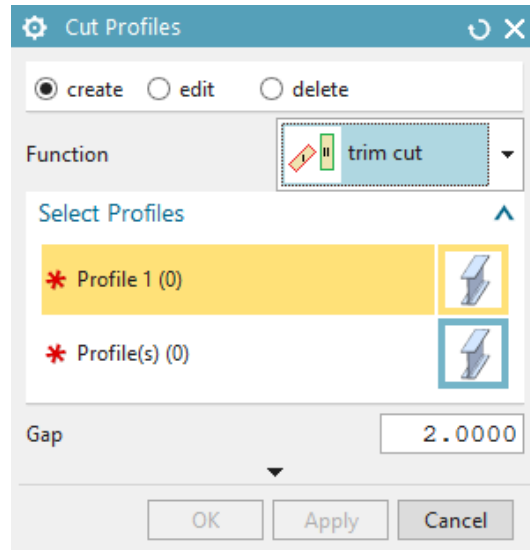
Both profiles are selected and ready to be cut off



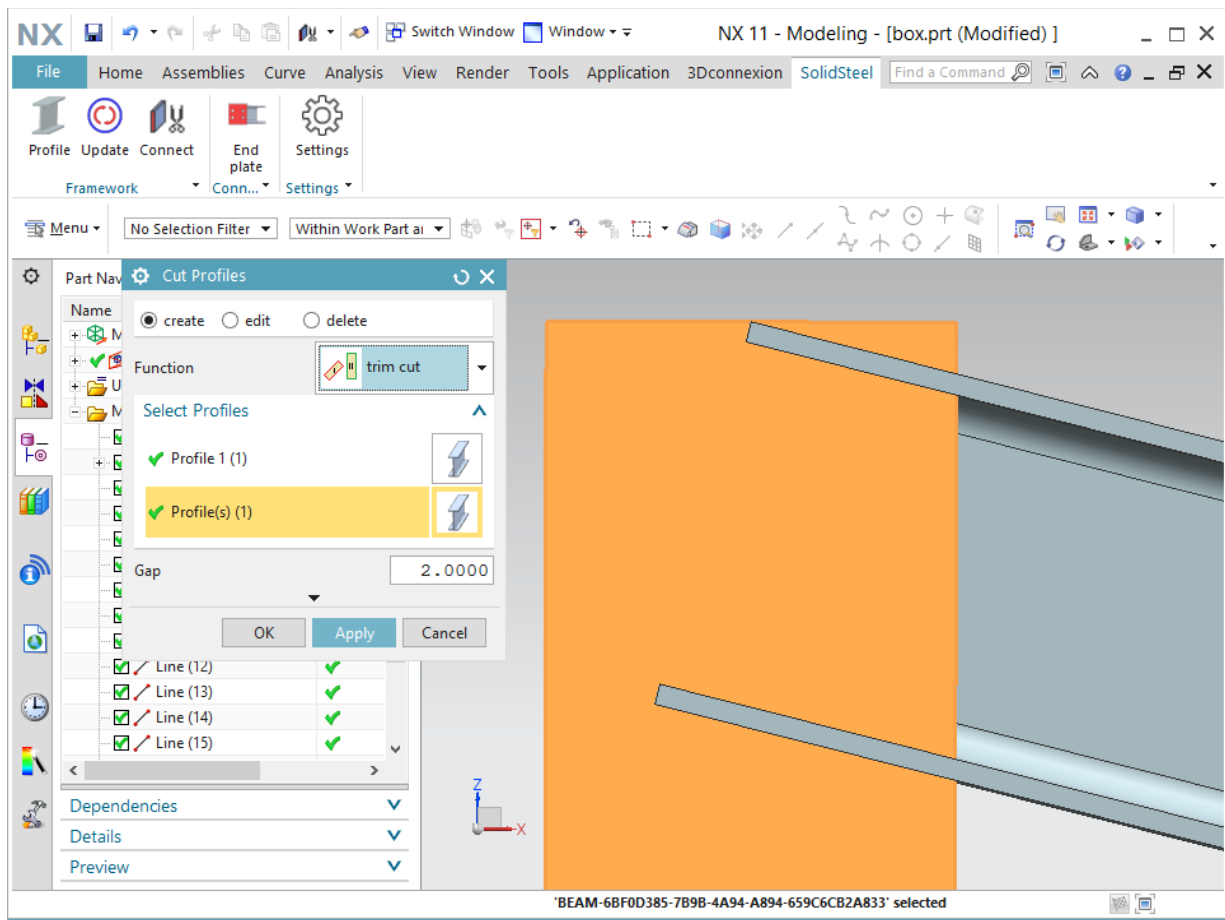
Result of the „cut“ function

trim cut

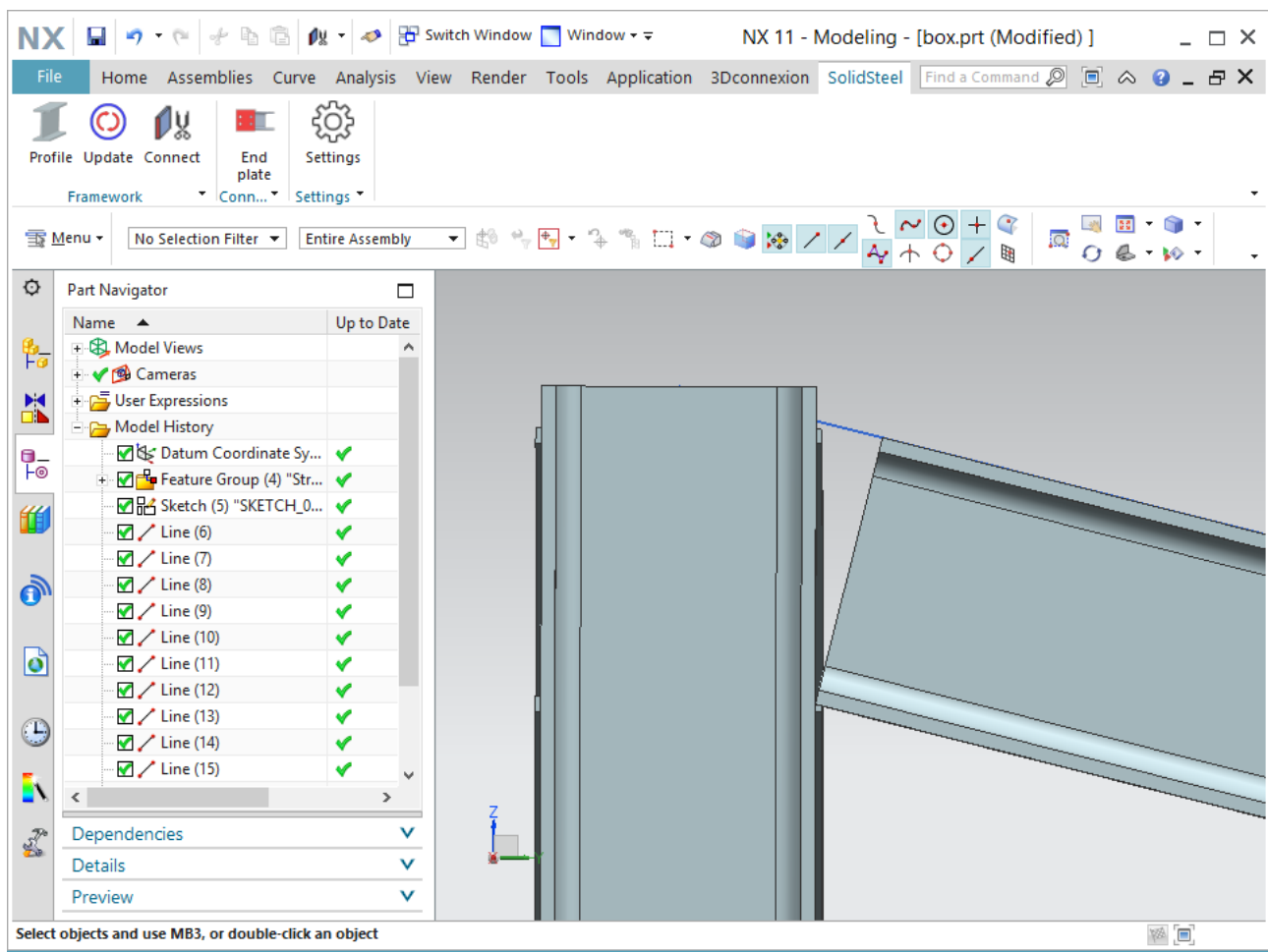
The trim cut function cuts the profile off with an angle of 90°. To do that select two profiles and click apply. If you need a gap between those two profiles, set the distance.



Cut profiles dialogue



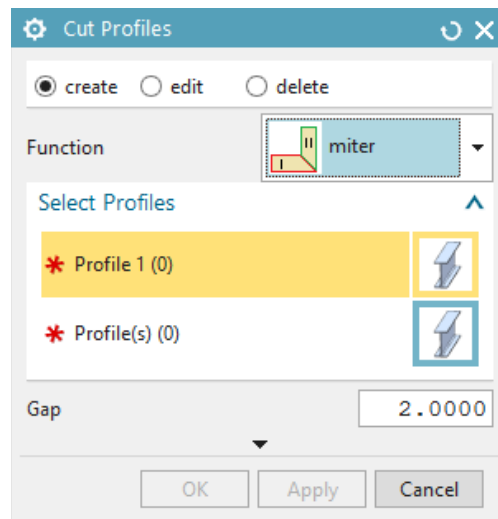
Both profiles are selected



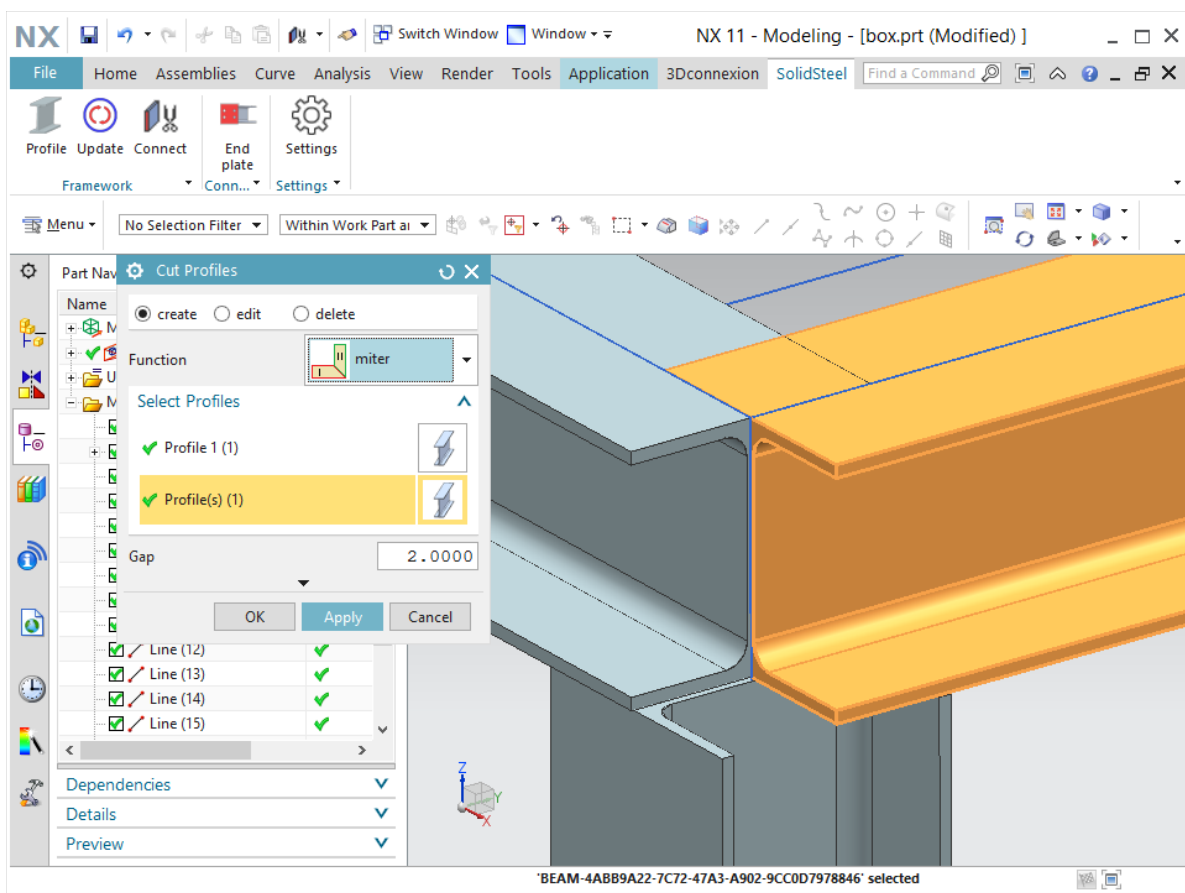
Result of the „trim cut“ function

miter cut

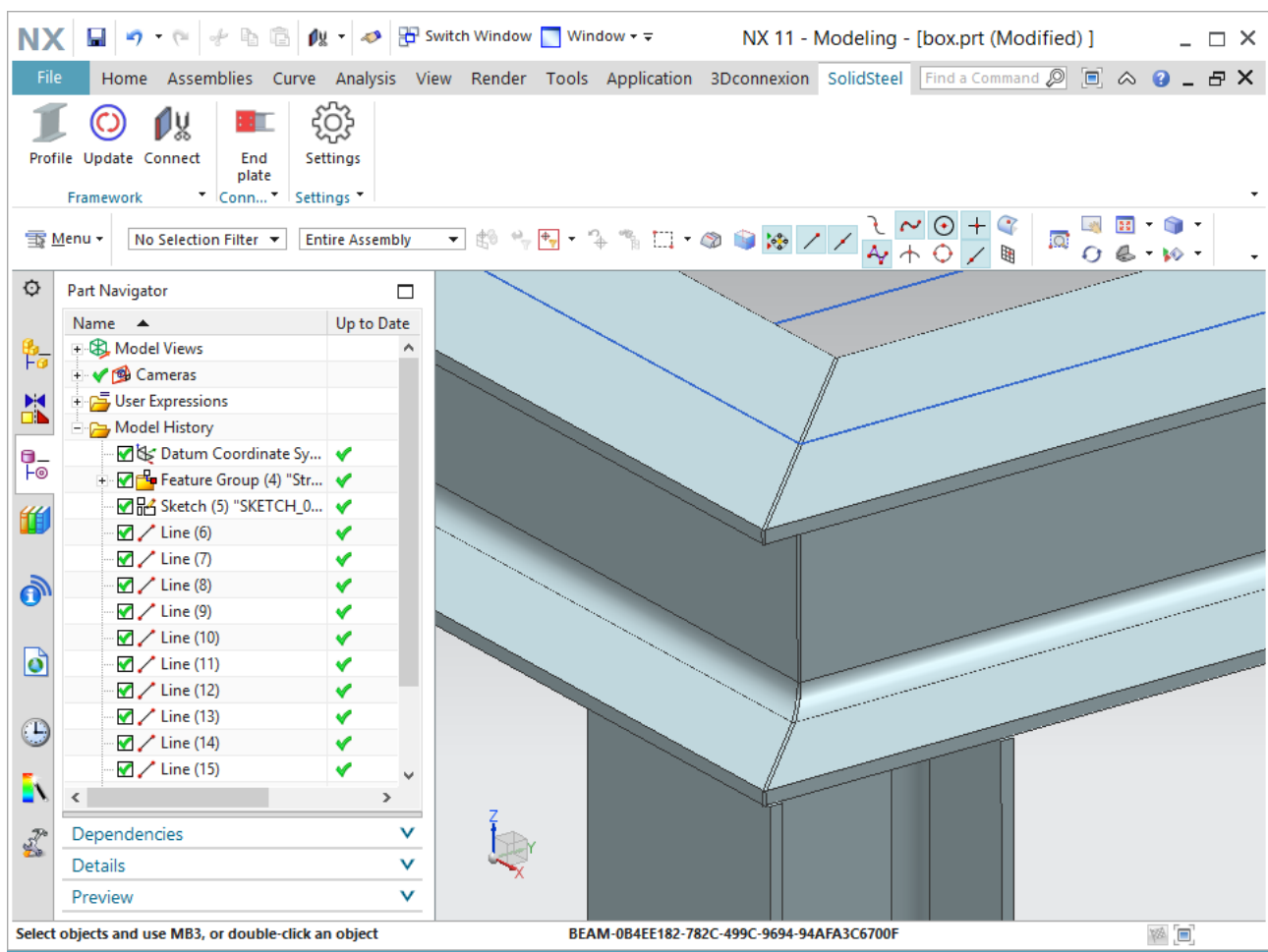
The miter cut function cuts two profiles off with an miter cut. If necessary, the profiles will be extended. To do that select two profiles and click apply. If you need a gap between those two profiles, set the distance.



Miter cut dialogue



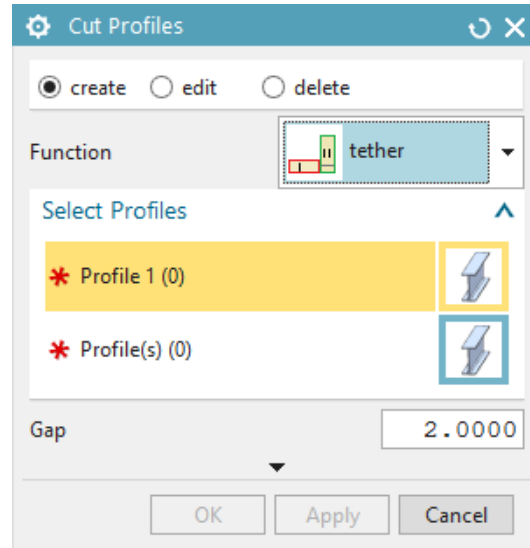
Both profiles are selected



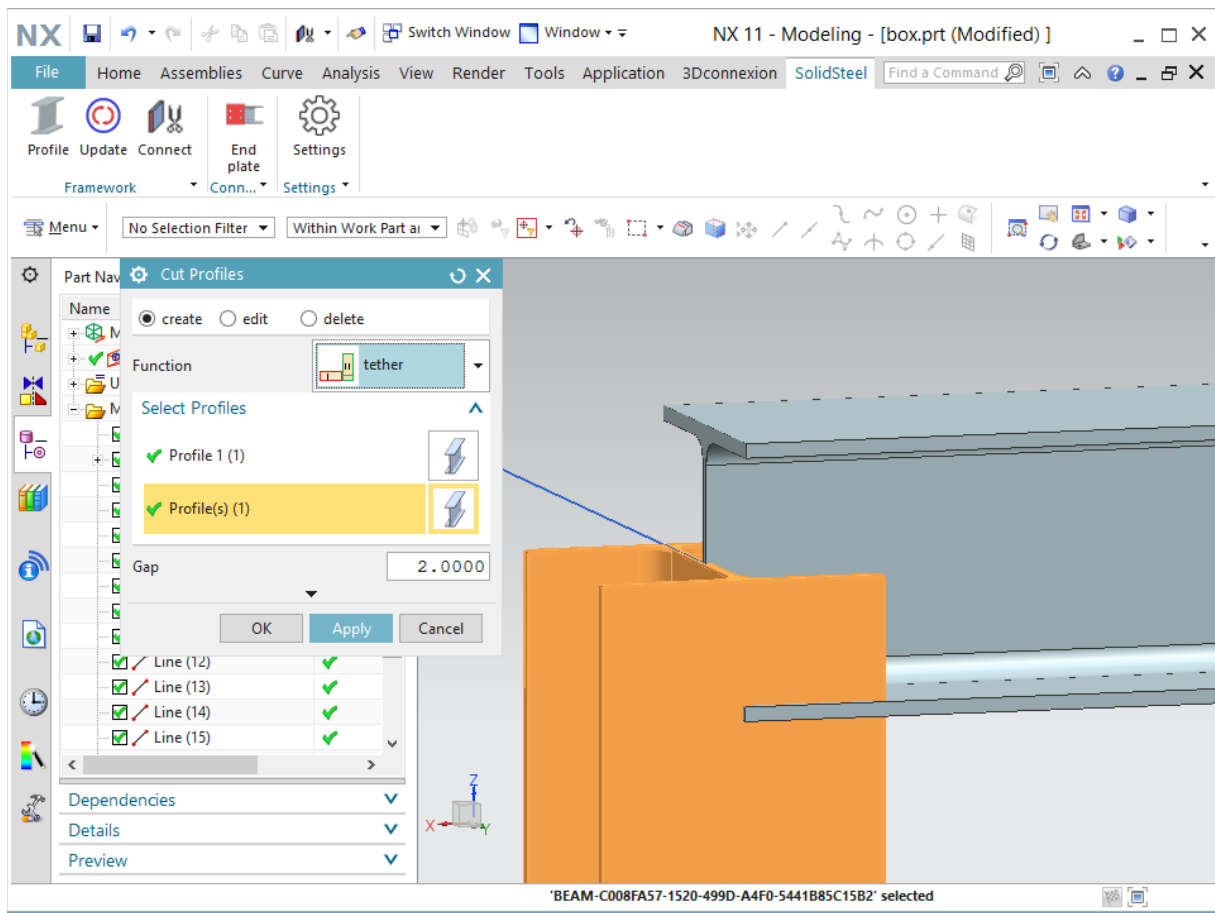
Result of the miter cut

tether cut

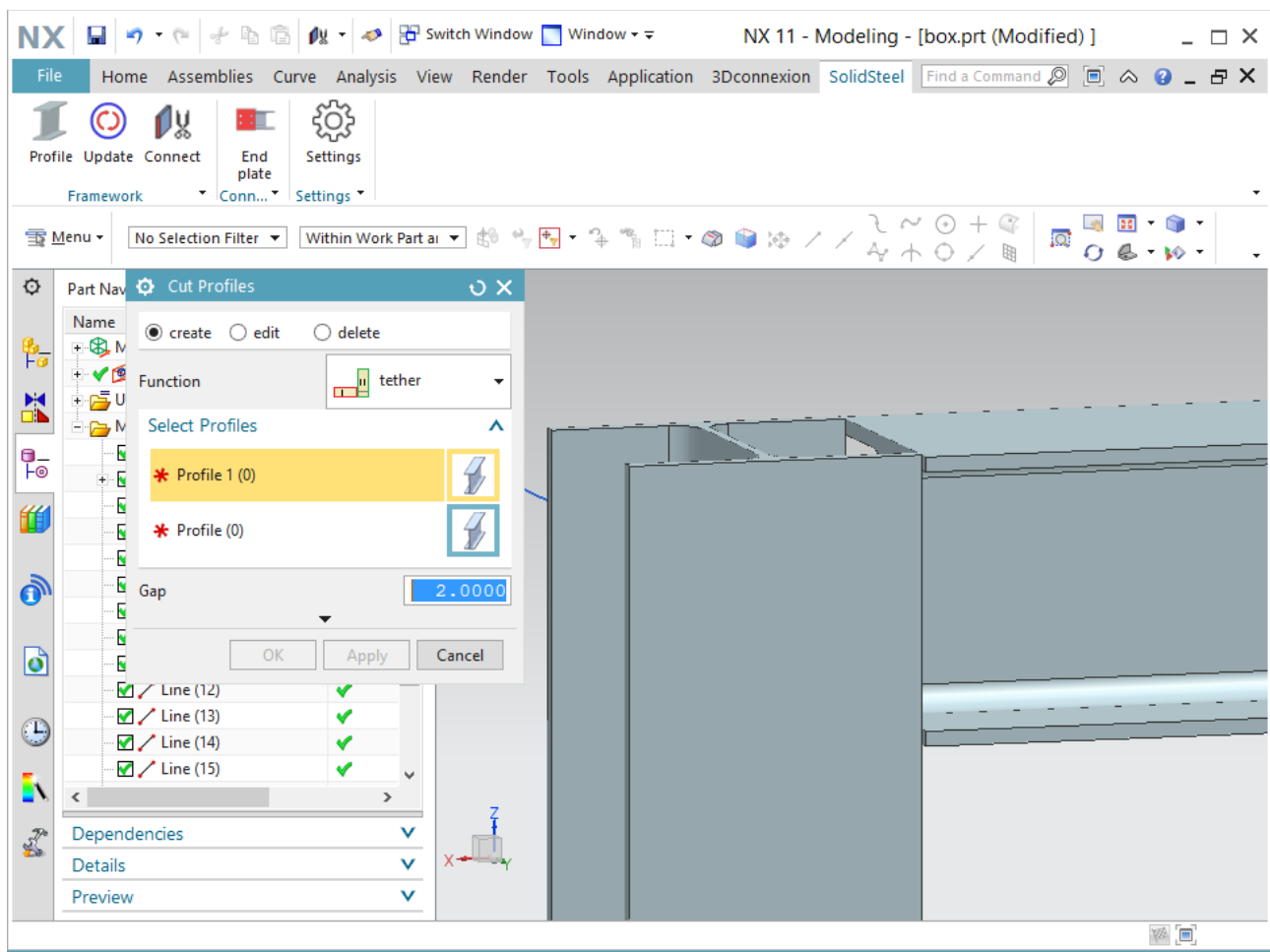
The tether cut function cuts the first profile off and the second profile will be extended. To do that select two profiles and click apply. If you need a gap between those two profiles, set the distance.



Tether cut dialogue



Both profiles are selected. The second one will be extended. The first one will be cut off.



The result of the tether cut

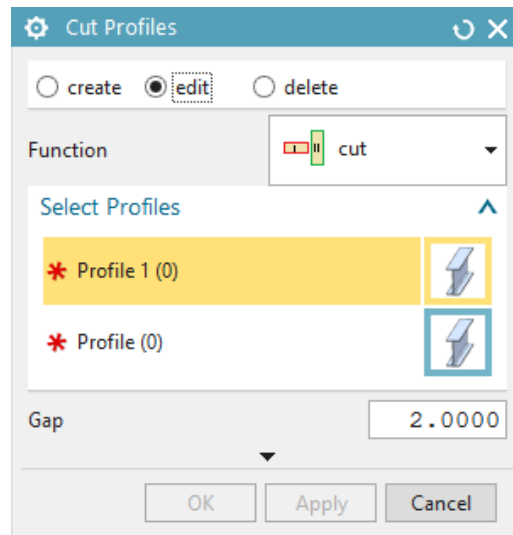
adjust cut

Not available today

The adjust cut functions are not available in this software version. The functions will be available within one of the next upcoming updates.

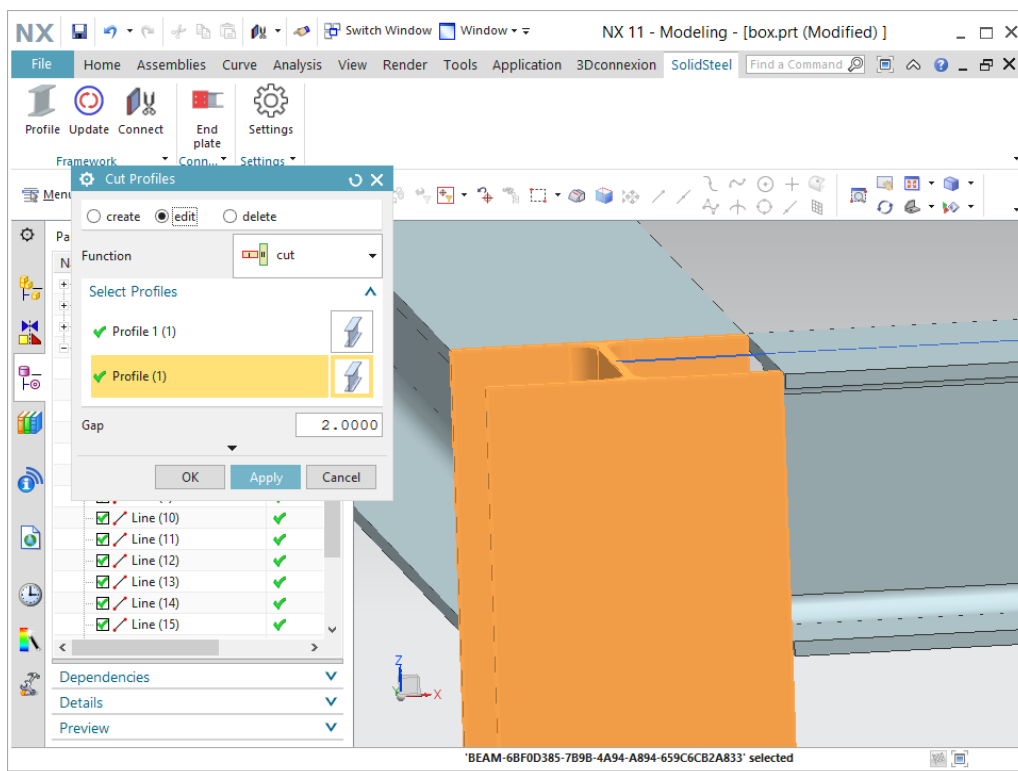
edit cuts

If an existing cut should be changed, SolidSteel parametric offers the possibility to do that. To edit an existing cut, just open the „cut profile“ dialogue and select „edit“ in the top selection menu.

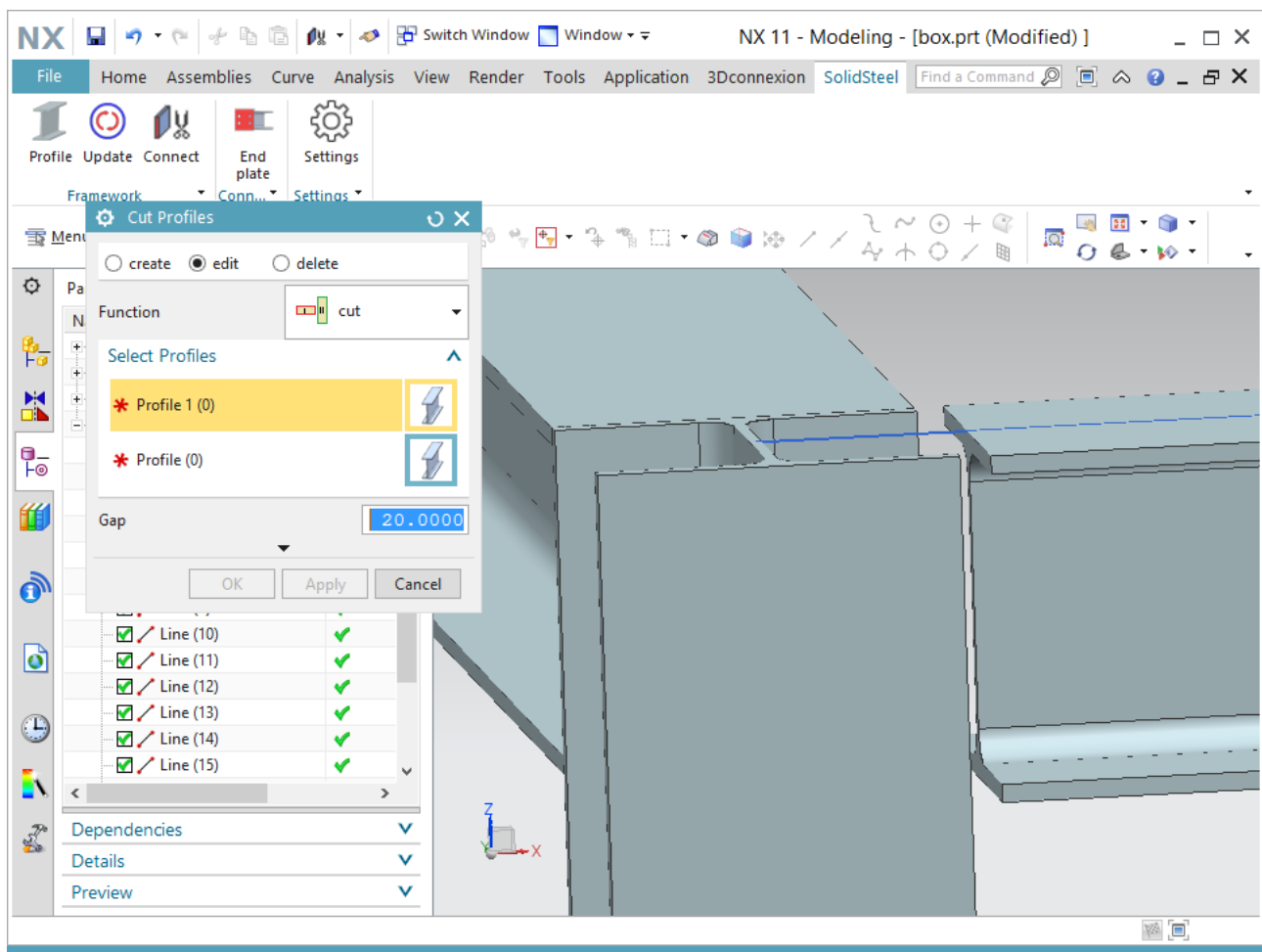


Editing an existing cut

After selecting both profiles, the existing settings will be loaded. Now you can change the gap or the cut function.



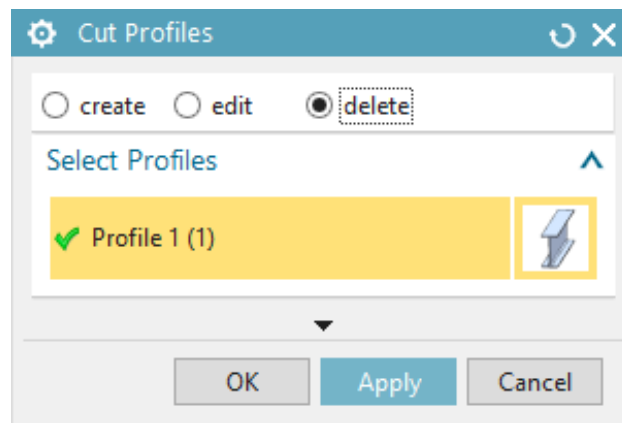
Both profiles are selected and the settings are loaded



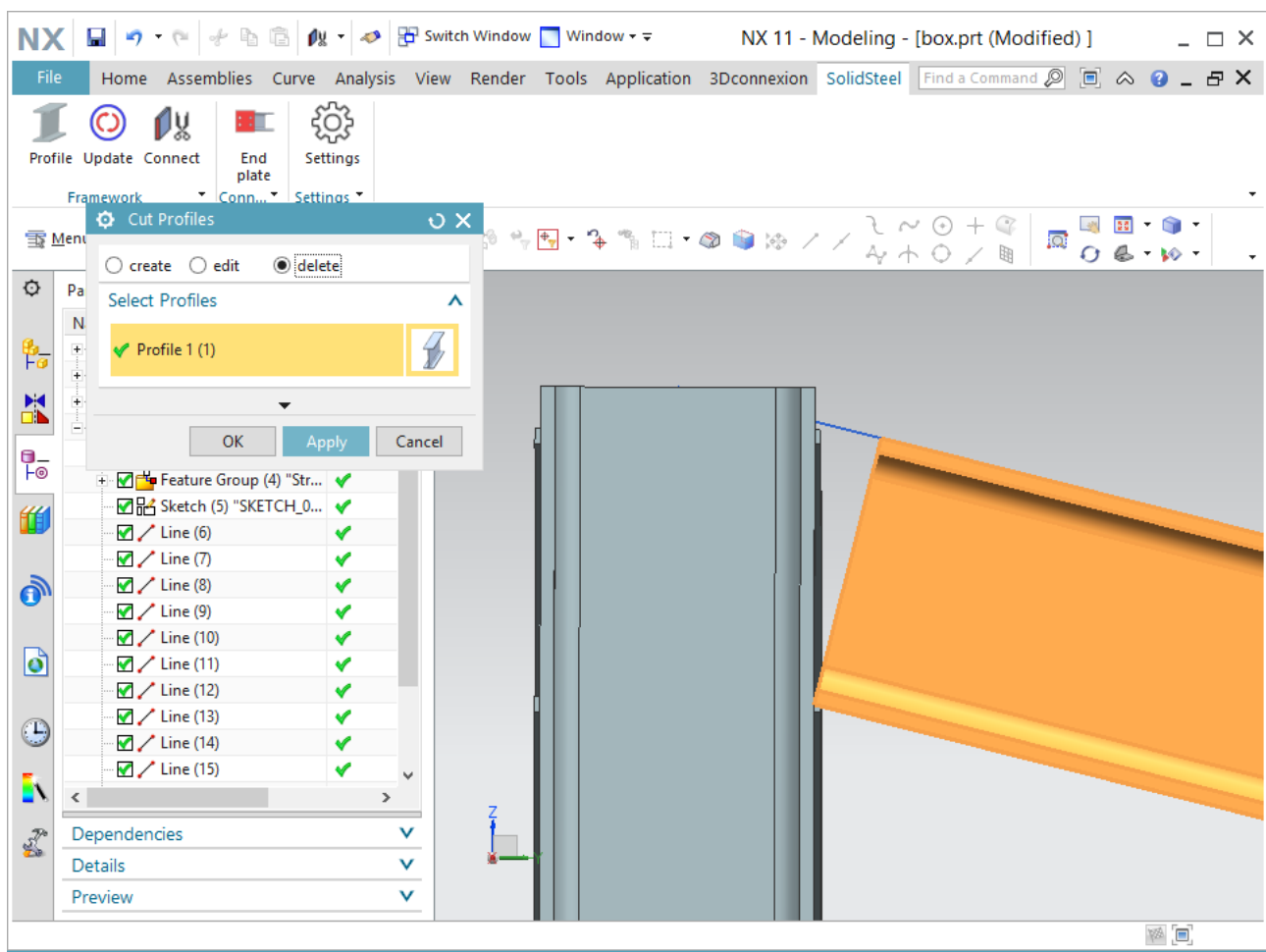
The gap between both profiles is changed

delete cuts

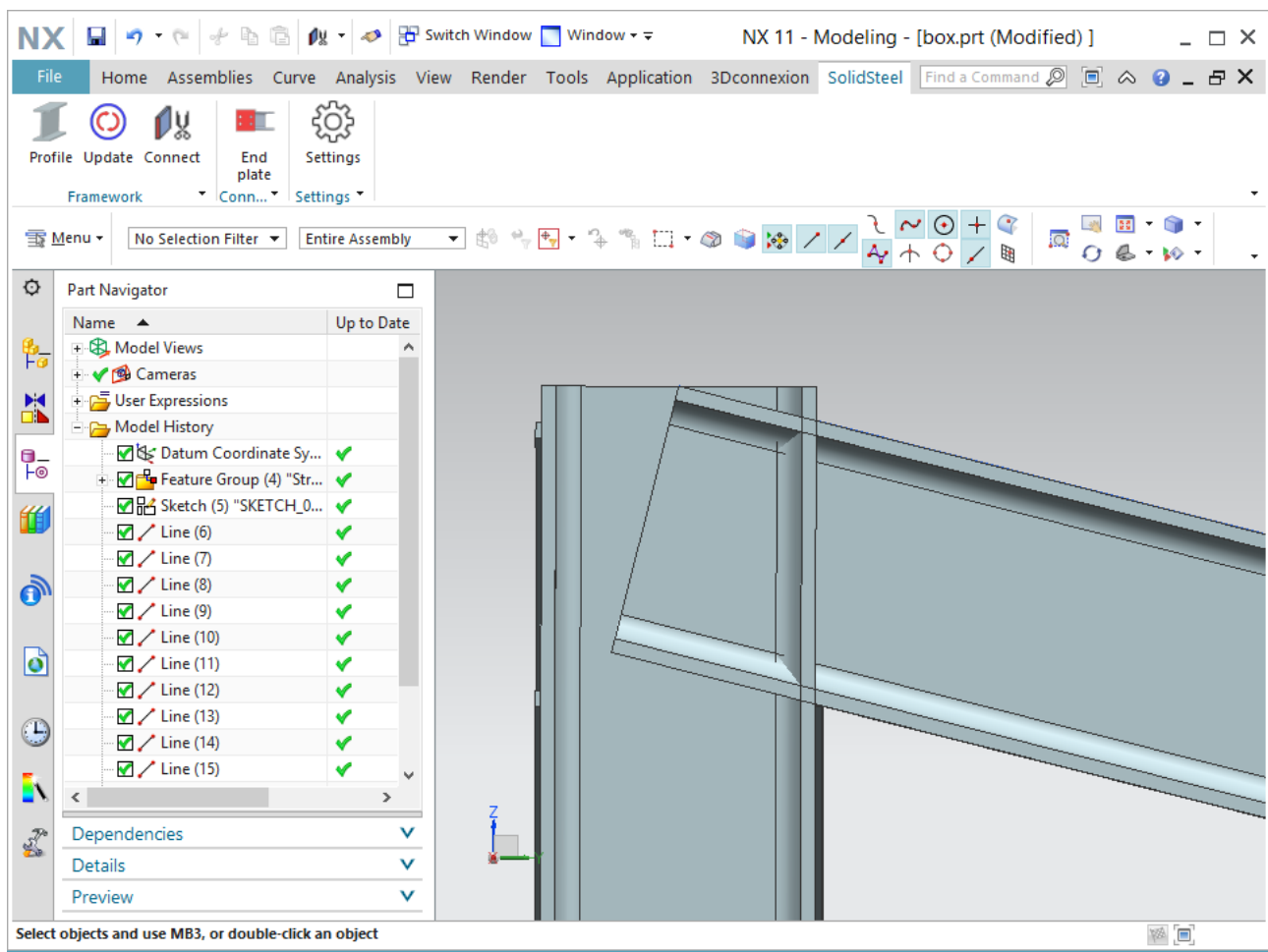
Existing cuts can be deleted. Just select the profiles and click apply. Then everything will be deleted.



Deleting and existing cut



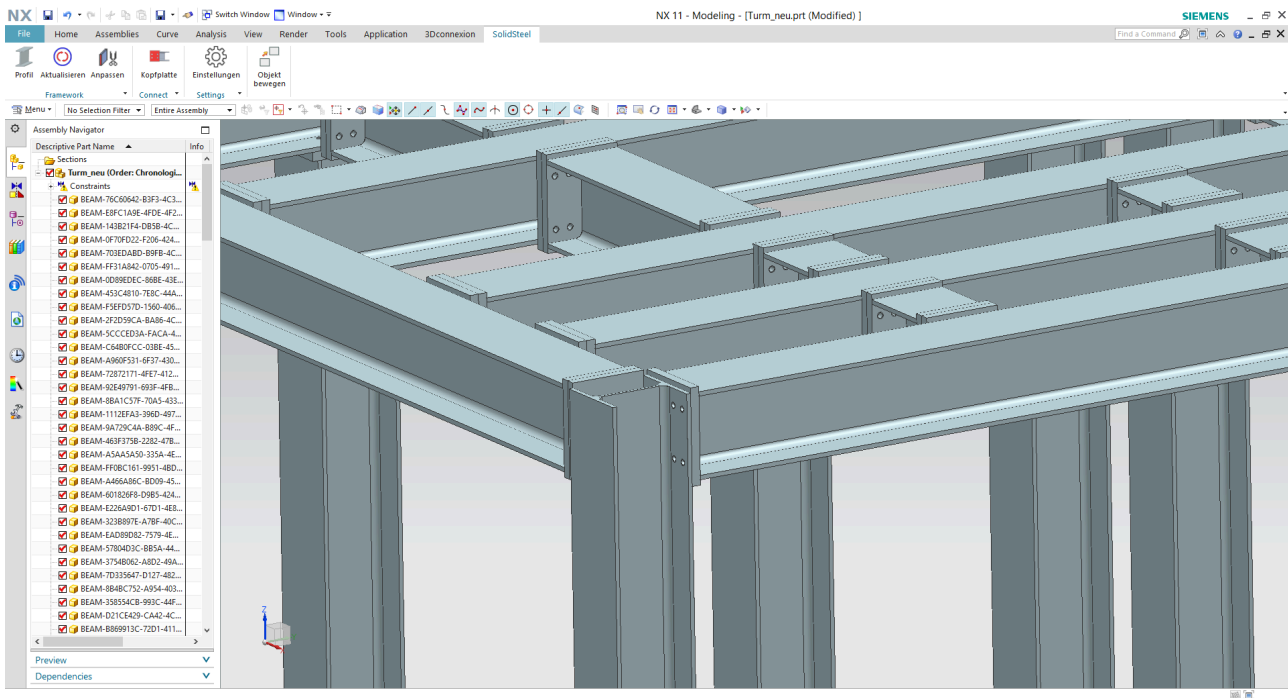
Situation before deleting the trim cut



Situation after deleting the trim cut

End plates (DAST)

End plates are important typical steel connections. In SolidSteel parametric for Siemens NX different types of end plates are available.



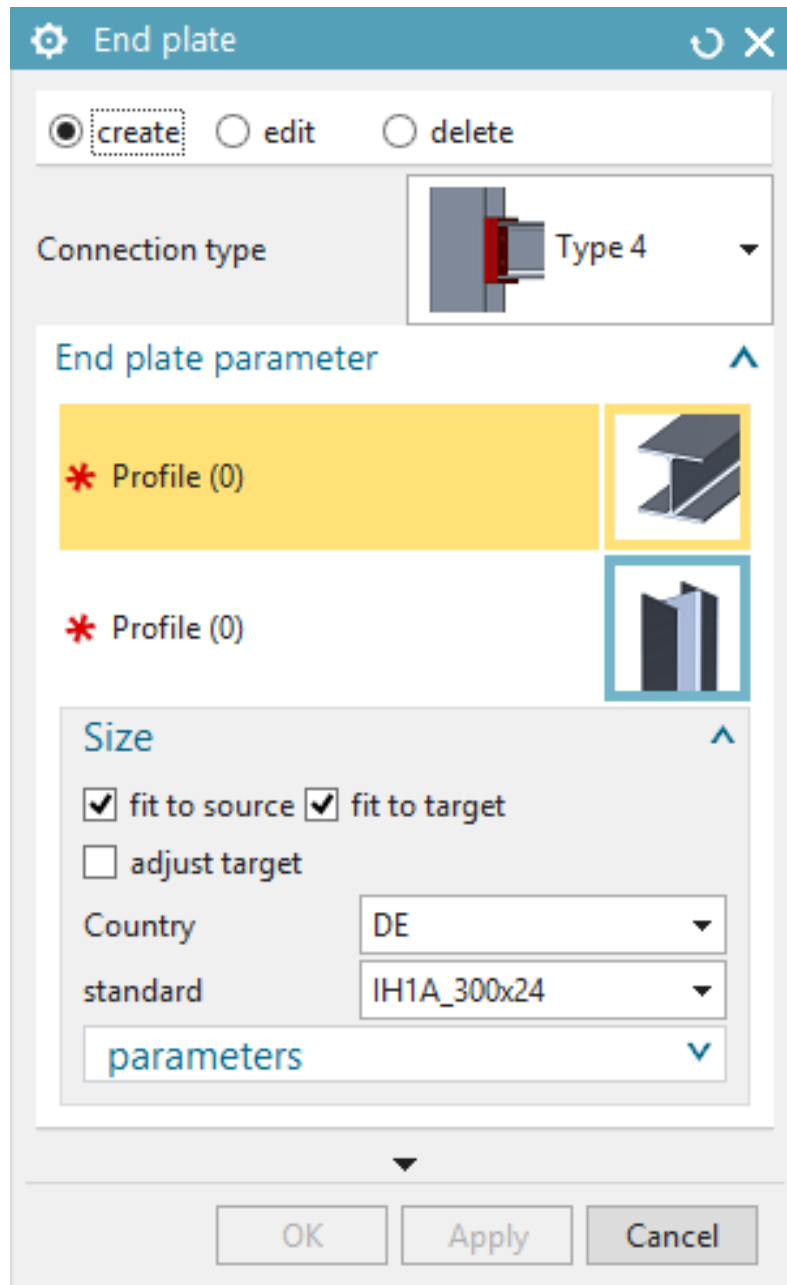
German standards only!!!

In the current version of SolidSteel parametric, the end plates are just working with german mm profiles. Then the german DAST standards will be used.

Other end plate standards and individual end plates will be coming soon in one of the following versions of SolidSteel parametric for Siemens NX.

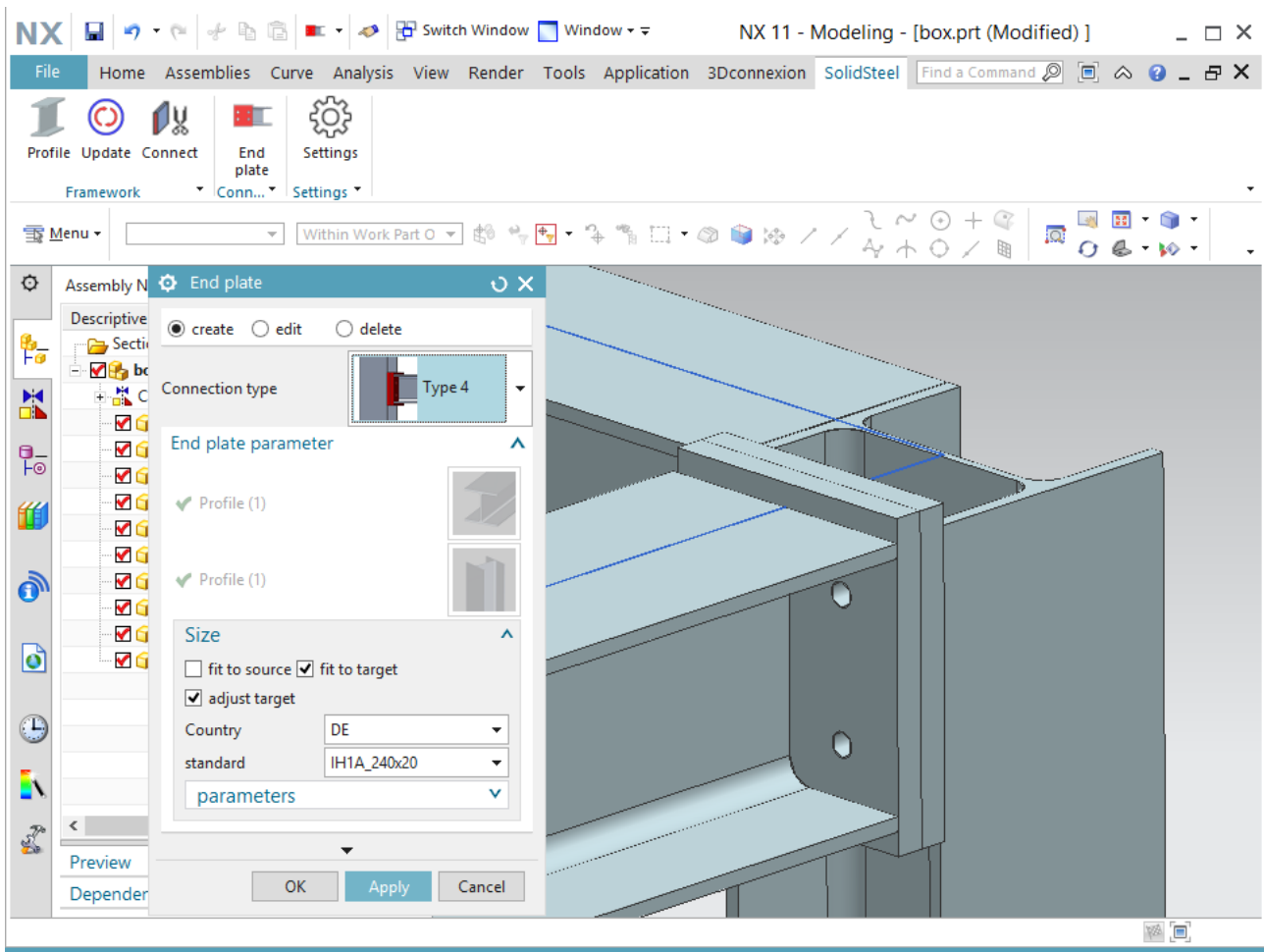
Placing end plates

For placing Endplates in SolidSteel parametric open the „end plate“ dialogue in the SolidSteel ribbon bar and go to „create“ in the top selection menu.



End plate dialogue - top selection: create

For placing an endplate, first select the connection type you want to use. Then select the two profiles you want with connect to each other.

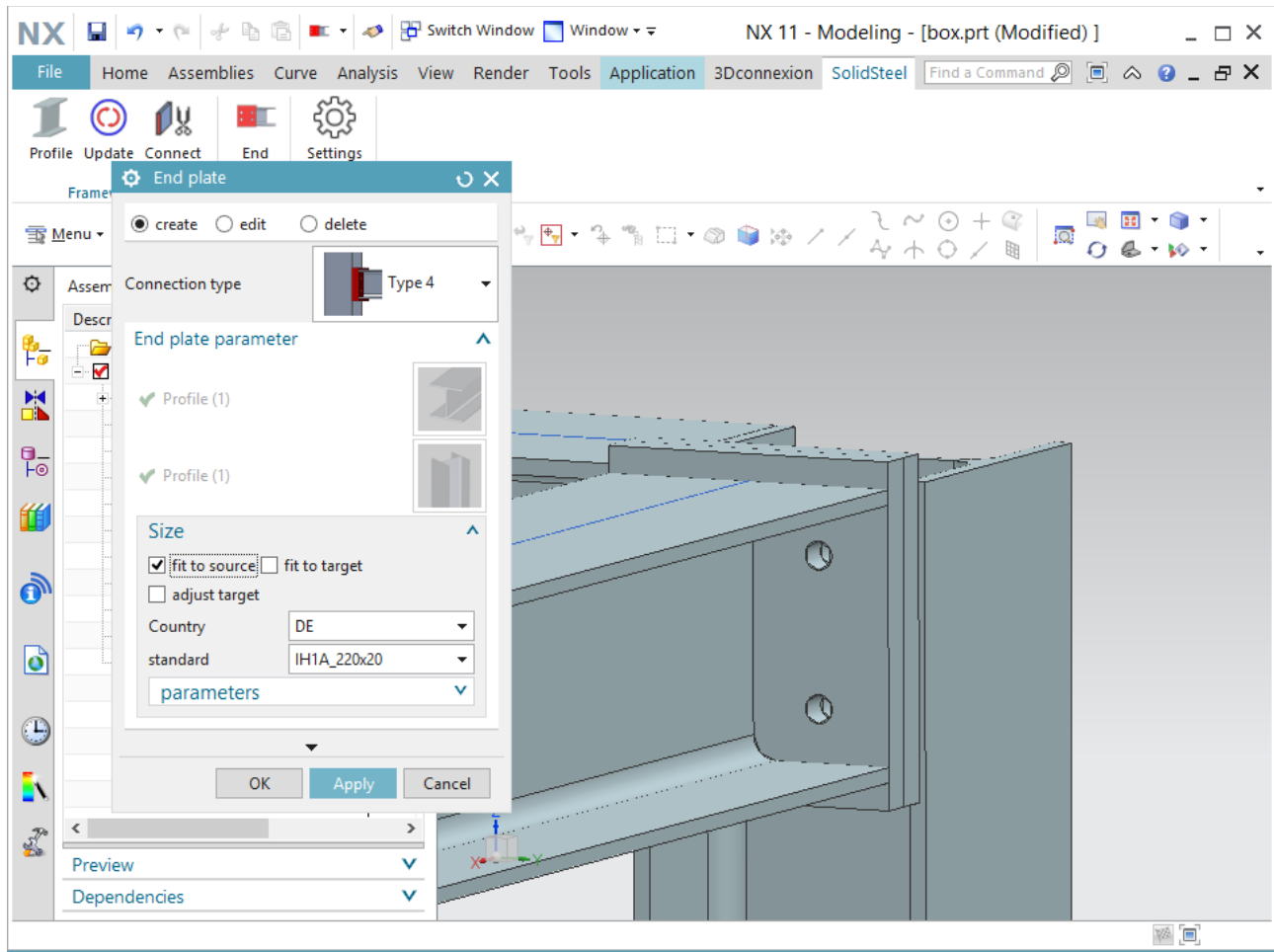


Placing an end plate connection with SolidSteel parametric for Siemens NX

After selecting the two profiles, the select option is disabled and a preview appears. The preview is one of the fitting german DAST end plate standards. Of course the standard can be changed. If you are happy with the connection, click apply.

The following chapters are about end plate connections in special situations.

fit to source

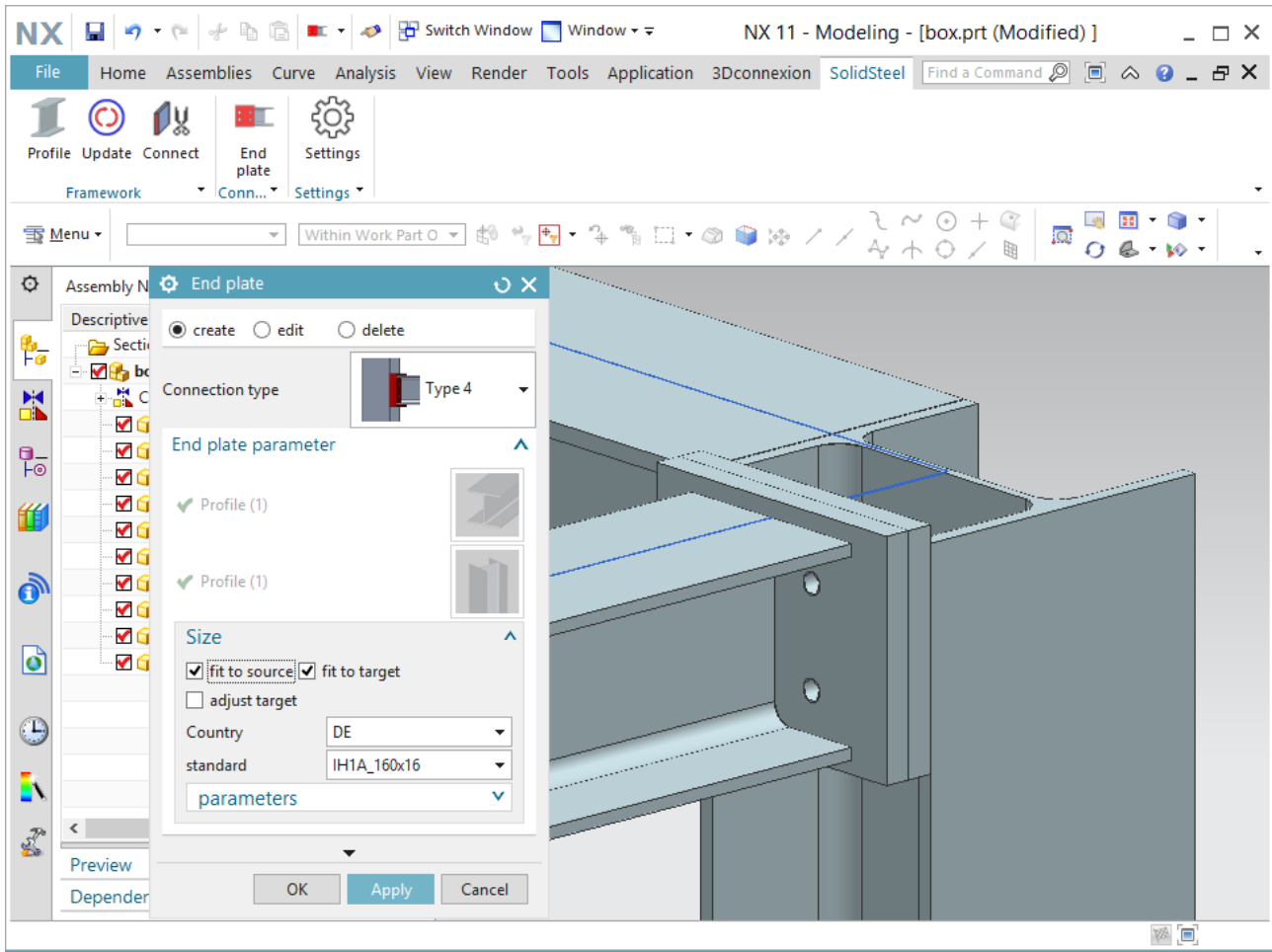


The end plate function „fit to source“ fits the endplate size for the profile size of the source profile.

End plate connection created with the „fit to source“ function

fit to target

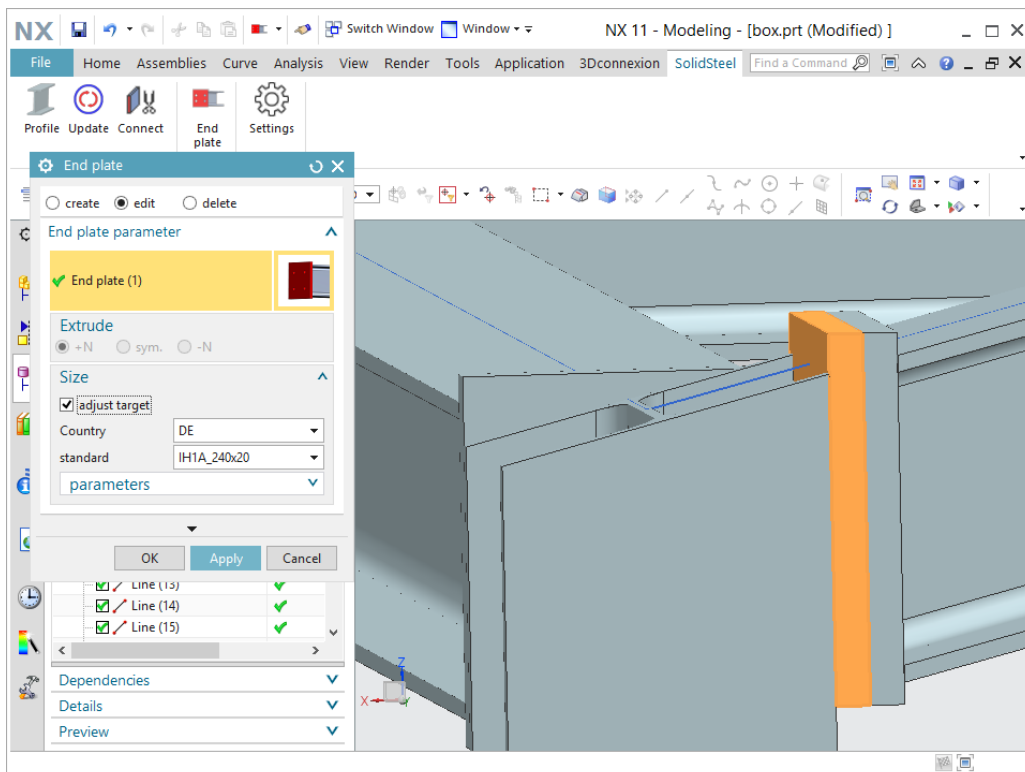
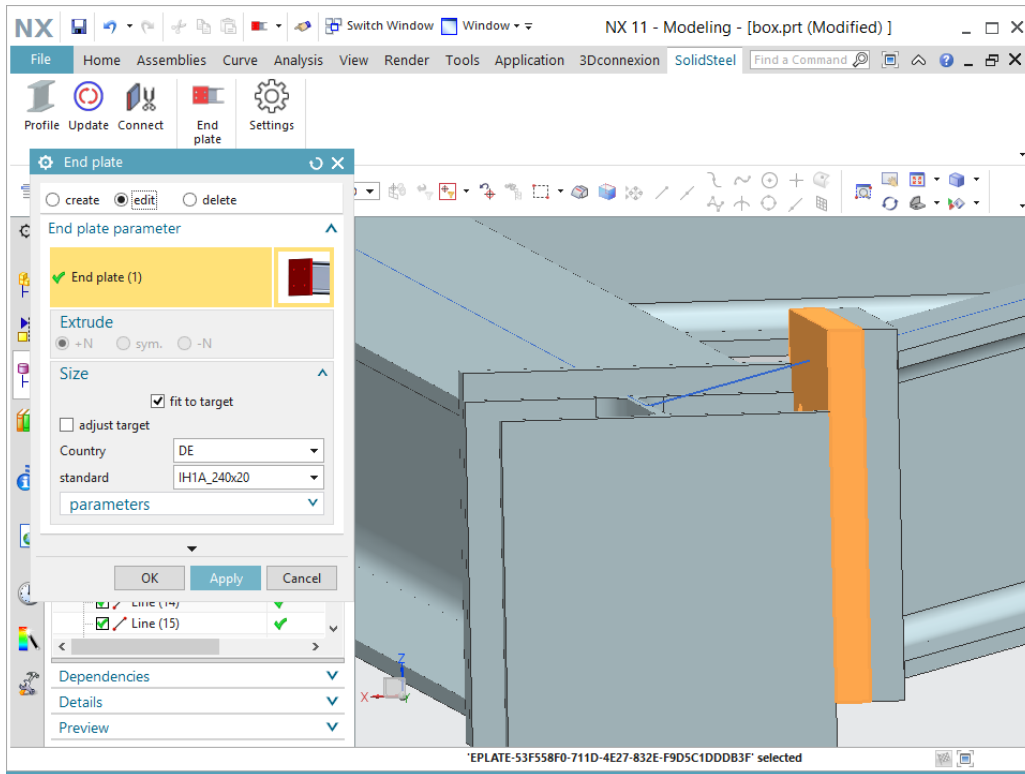
If you connect a small profile to a bigger profile you can use the fit to target function. This functions changes the size of the end plate to the size of the target profile.



End plate connection with „fit to target“ function

adjust target

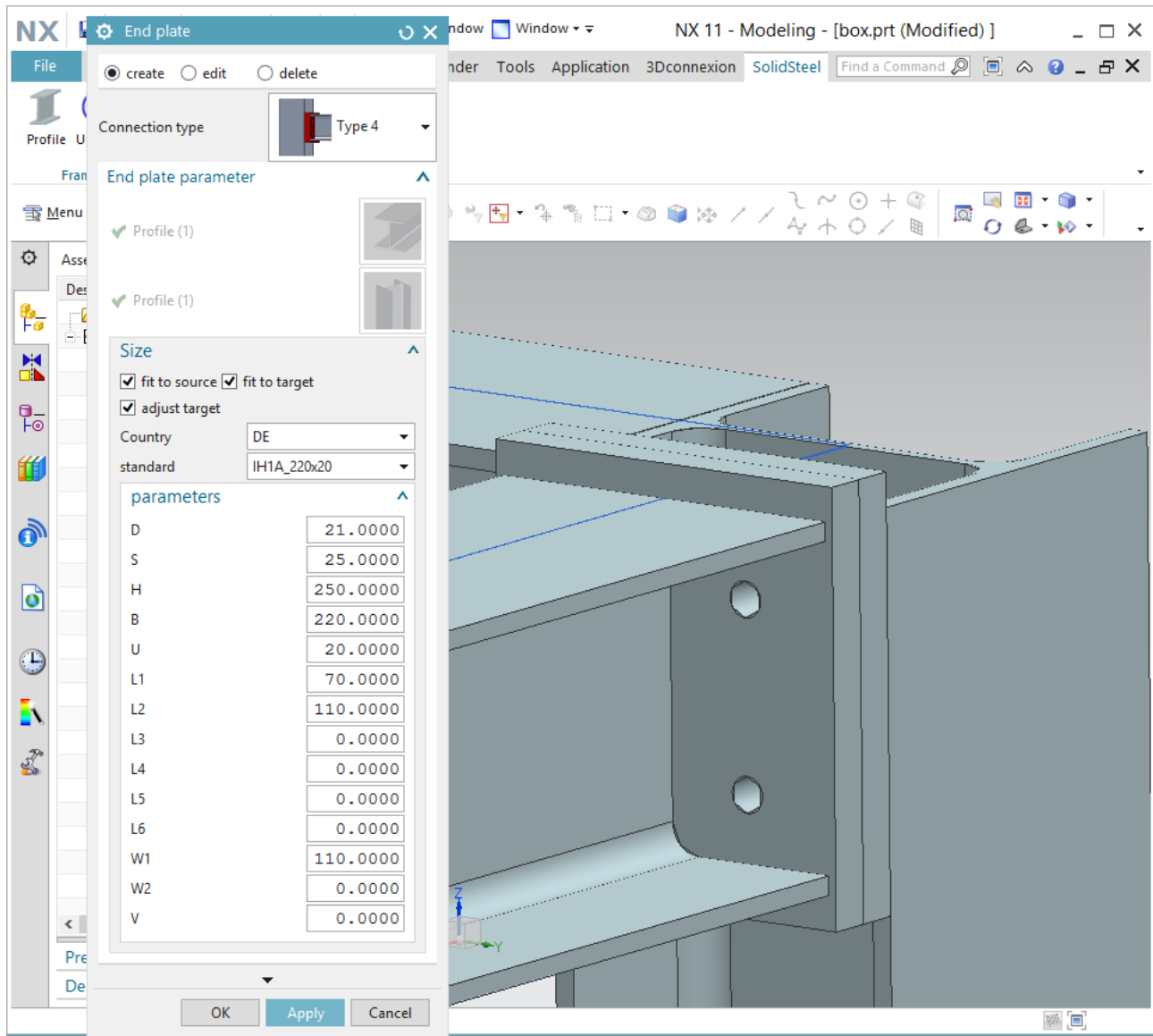
If the profiles of the end plate connection are not in one plane, you can use the „adjust target“ function. Then the target profile will be cut off in the plane of the source profile.



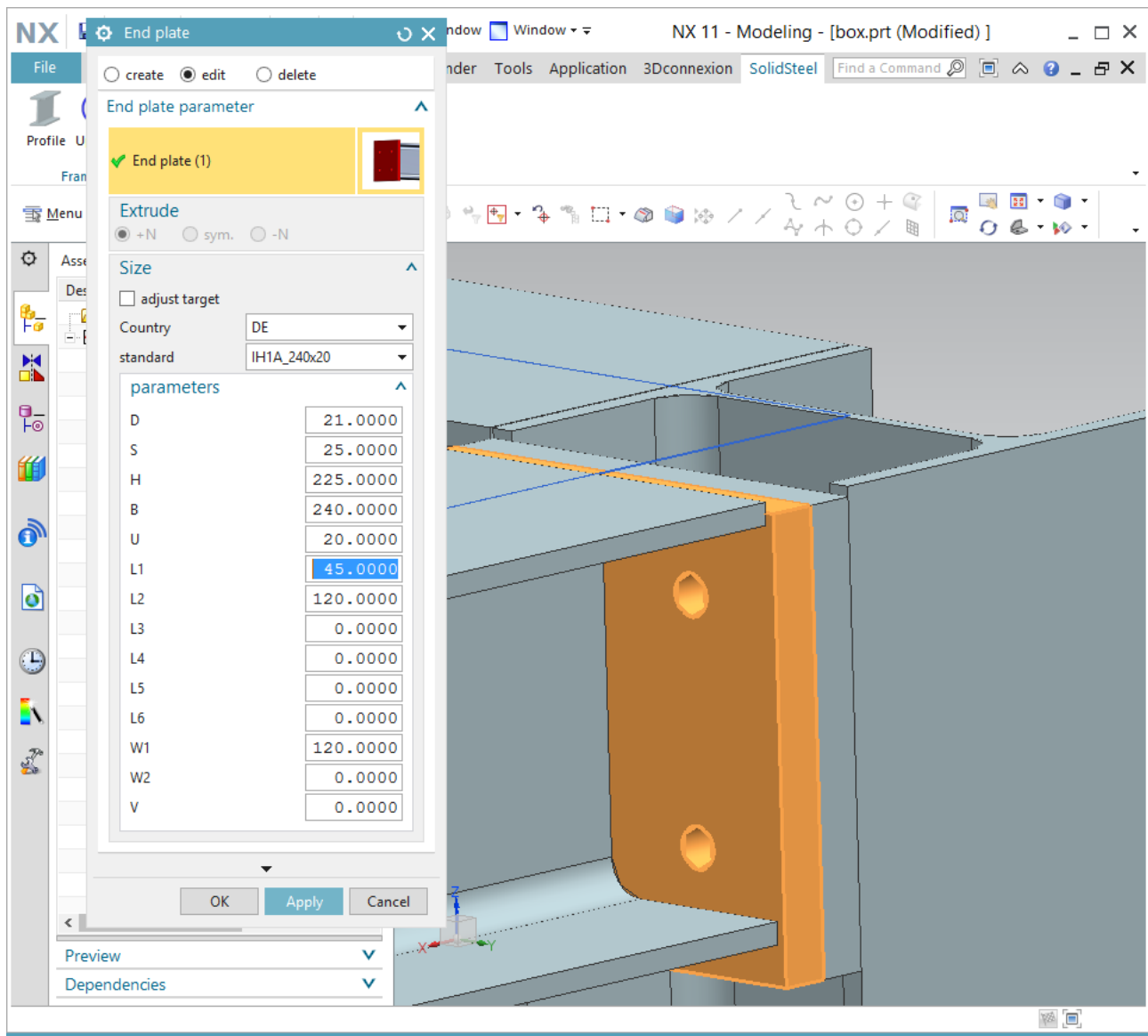
End plate connection with the „adjust target“ function

Changing the parameters

The parameters of the end plates can be changed if necessary. To do that, open the parameters menu. If you hold the mouse on the text „parameters“ a sketch appears.



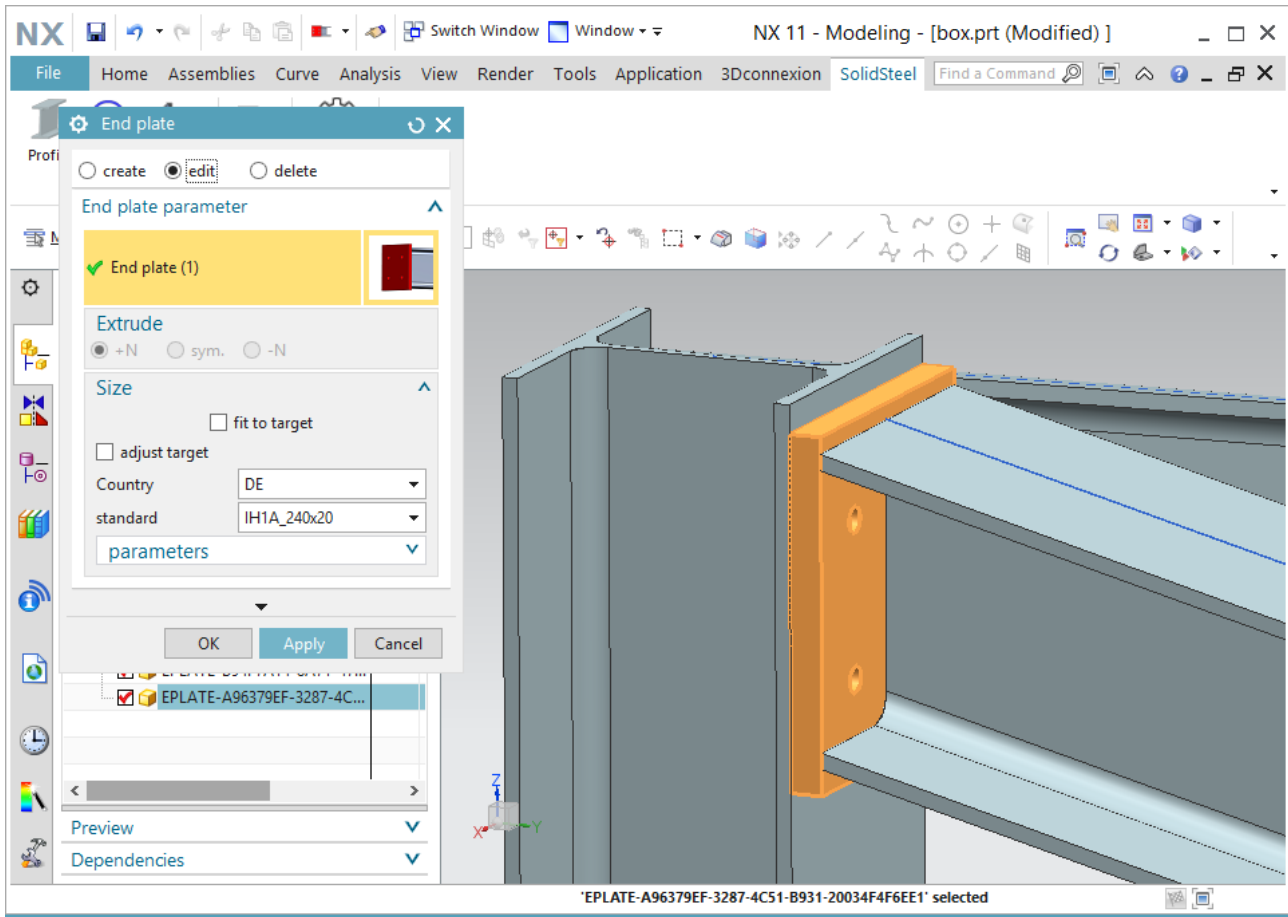
The possibility to change the parameters of the standard end plate connection



Modified DAST end plate connection

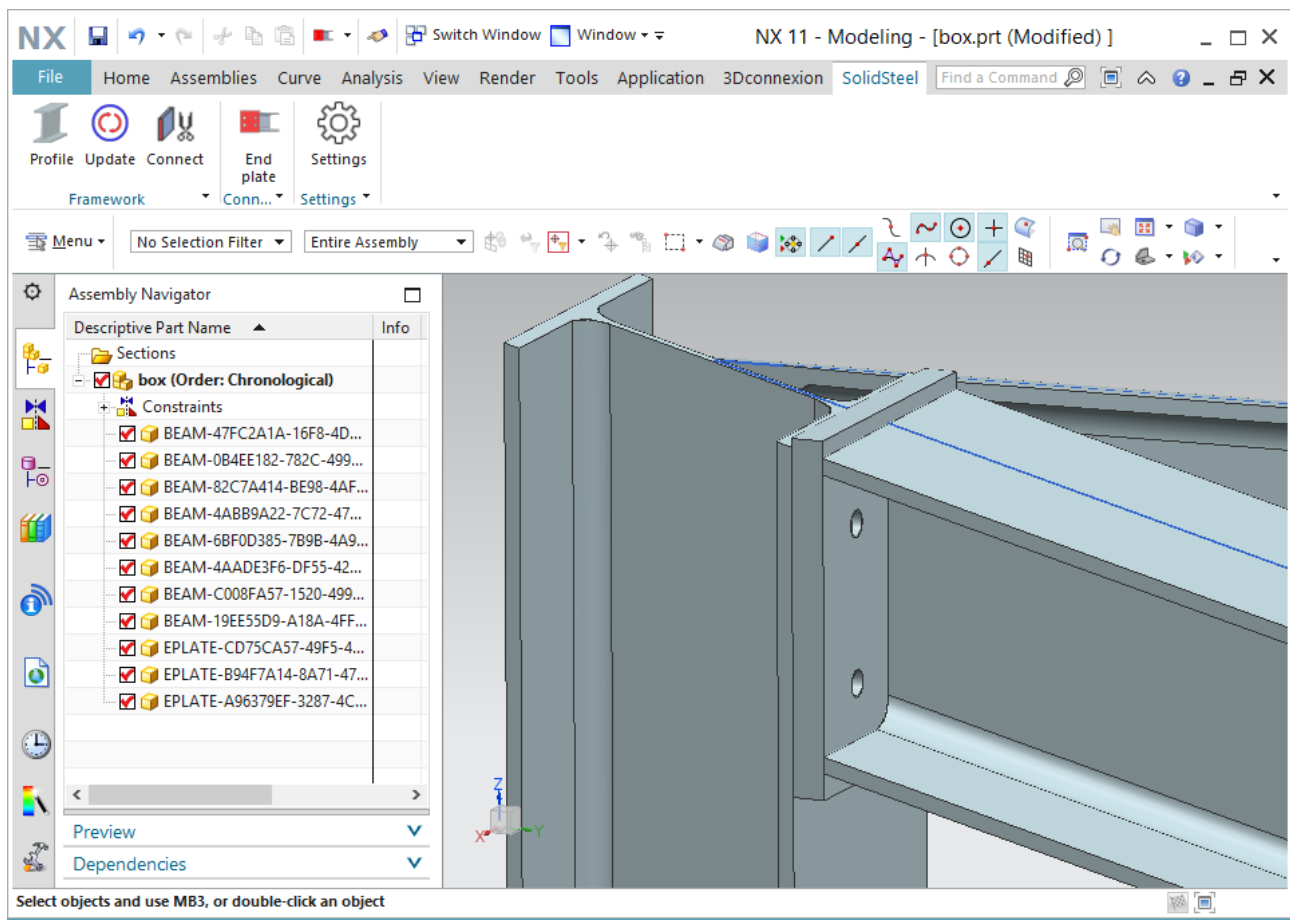
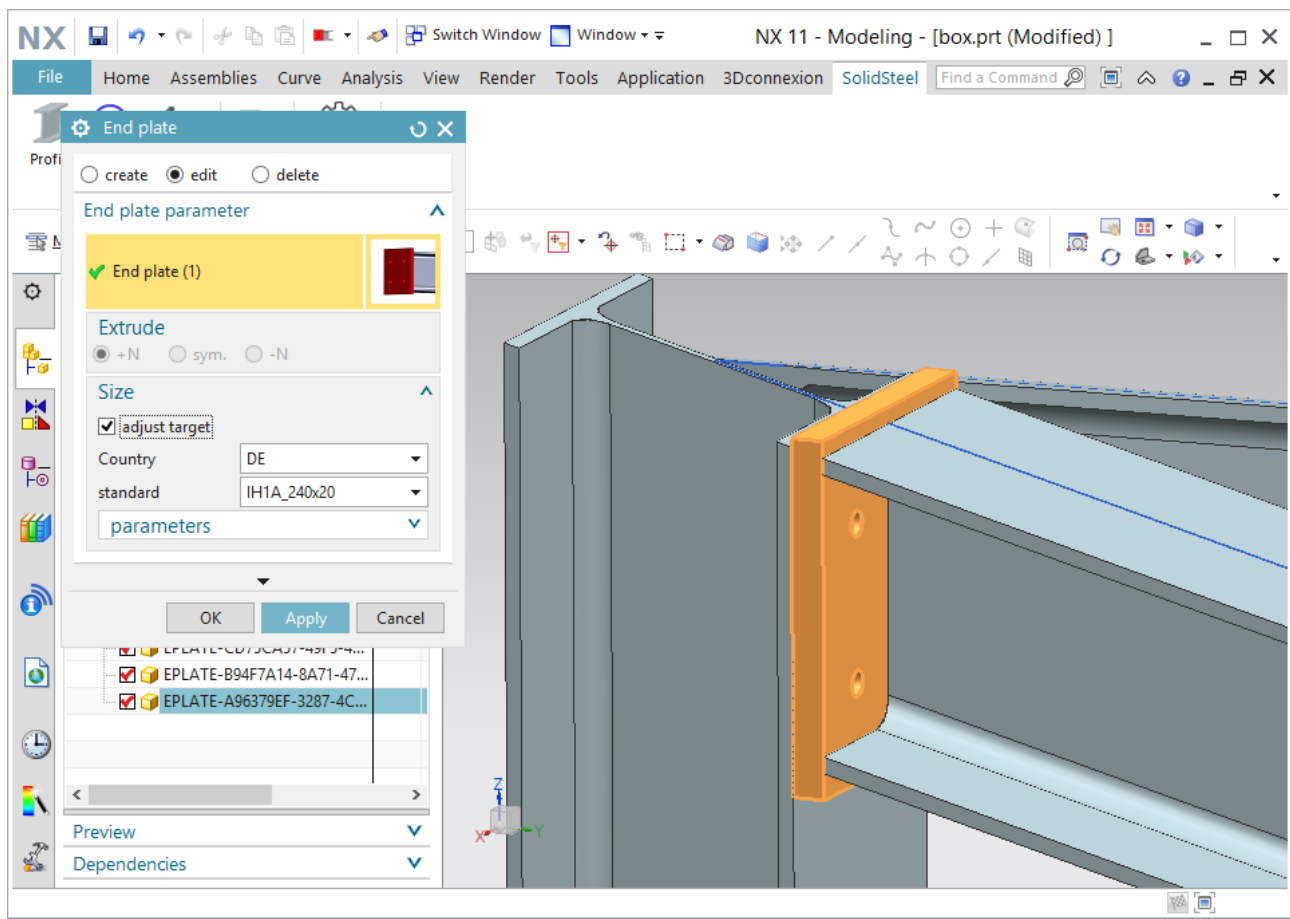
Changing end plates

Of course, existing end plates and end plate connection can be changed. To do that, open the „end plate“ dialogue and go to „edit“ in the top selection menu.



Editing existing end plate connections

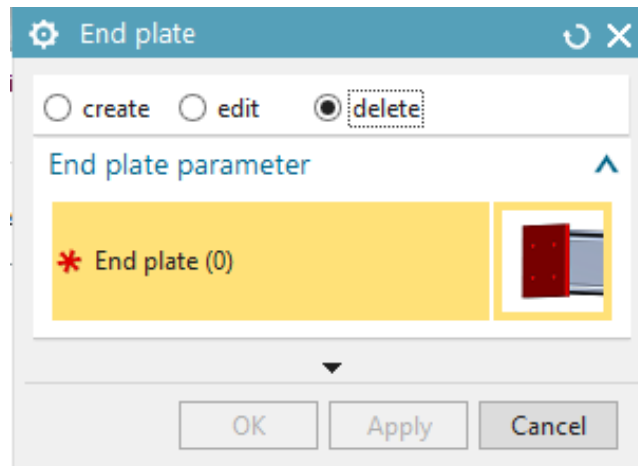
At first, select the end plate connection you want to edit by clicking one of the end plates. Now all the settings are loaded and everything can be changed.



A changed end plate connection „Type III“

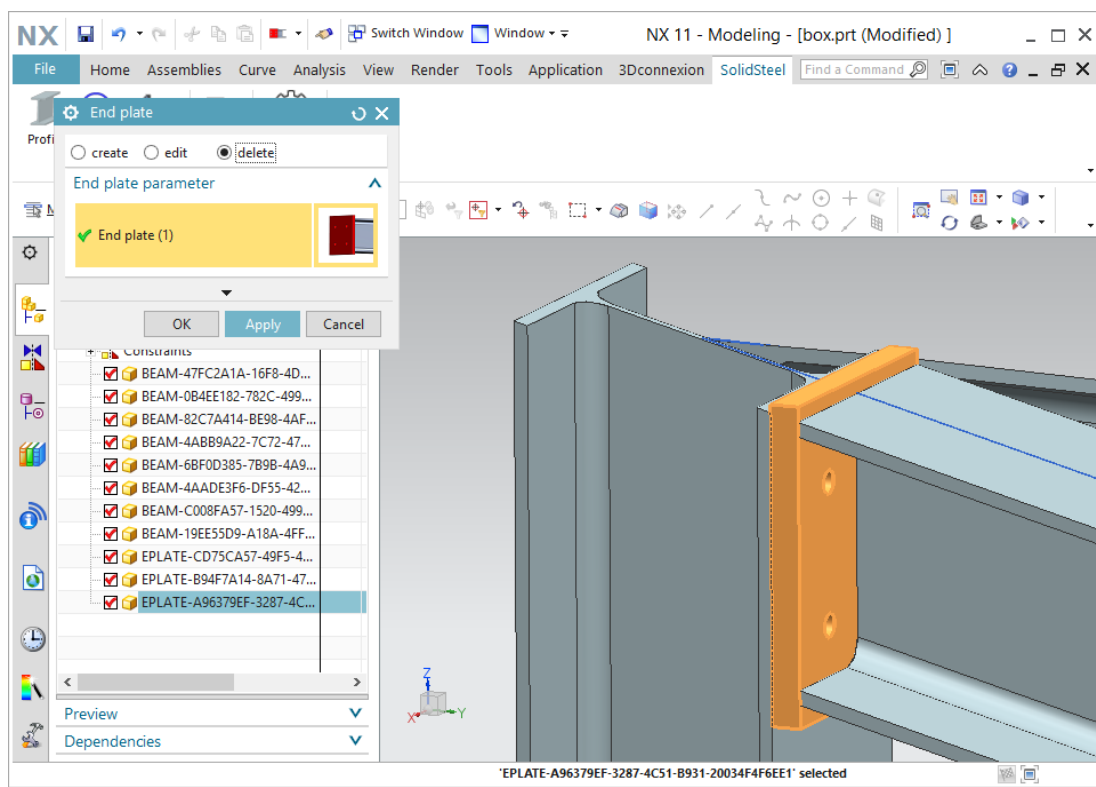
Deleting end plates

For deleting existing end plates and end plate connections, go to the „end plate“ dialogue and select „delete“ in the top selection menu.



Deleting an existing end plate or end plate connection

To delete an end plate or an end plate connection, select one of the end plates and click apply. All the profiles and changes will be cancelled.



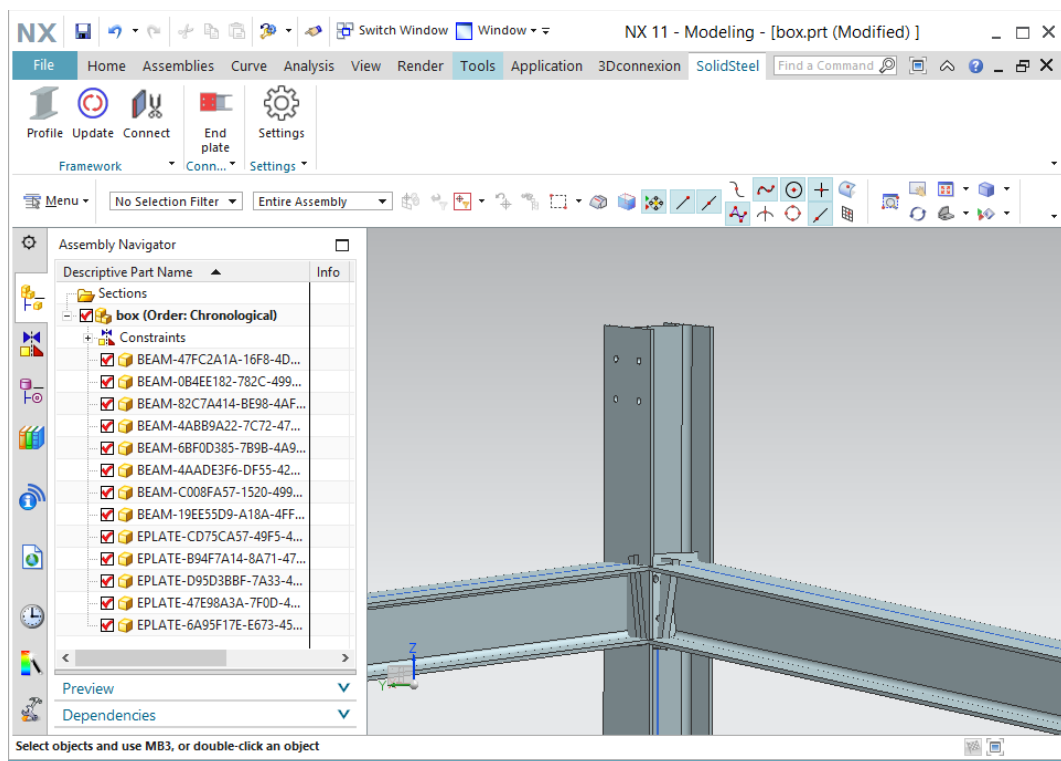
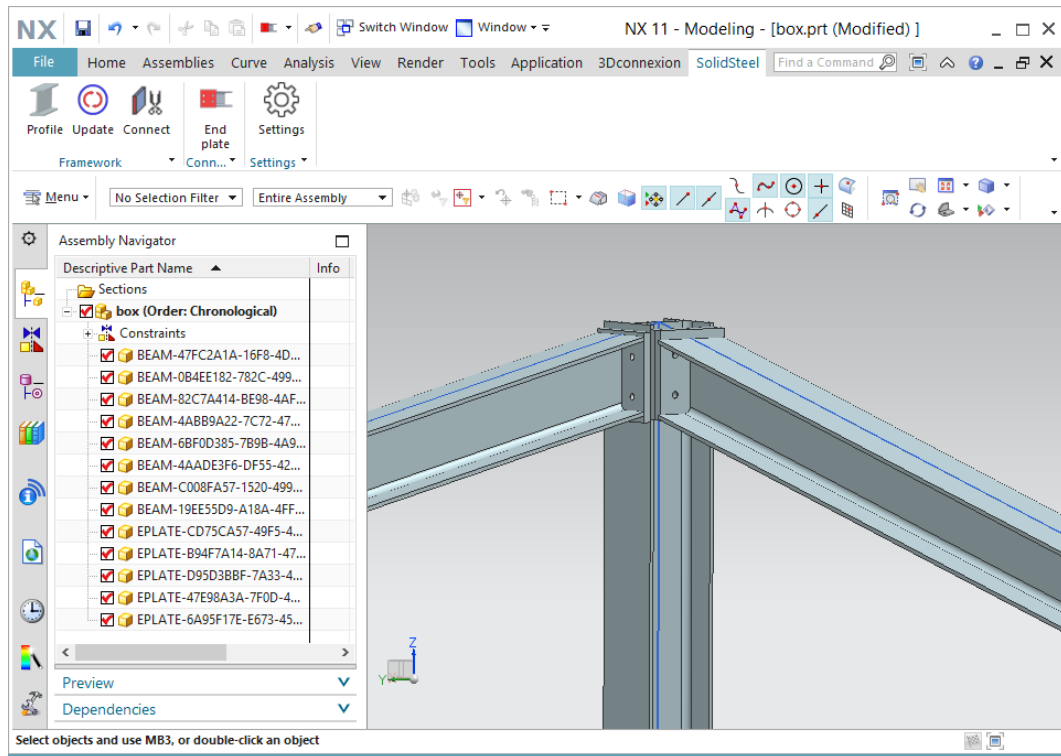
Selected the end plate which should be deleted

Major changes

As the name suggests, SolidSteel parametric is a parametric system for structural steel design. So if the sketch is changed, all the components including the steel connections will be changed, too.

Changing the framework

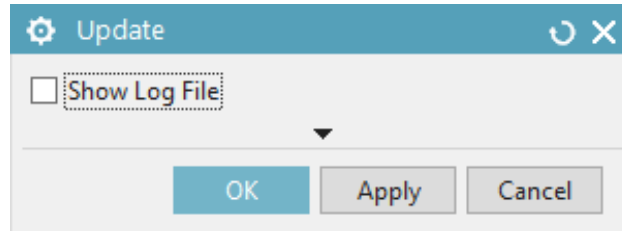
To change the framework, just change the Siemens NX sketch where the framework is connected to. All the profiles and connections will follow these changes.



Changed framework in SolidSteel parametric for Siemens NX

As you can see, the profile length, the angle of the connections and things like that are not updated, yet. In SolidSteel parametric is a special function for that.

To update the rest of the framework, go to the „update“ dialogue in the SolidSteel ribbon bar.

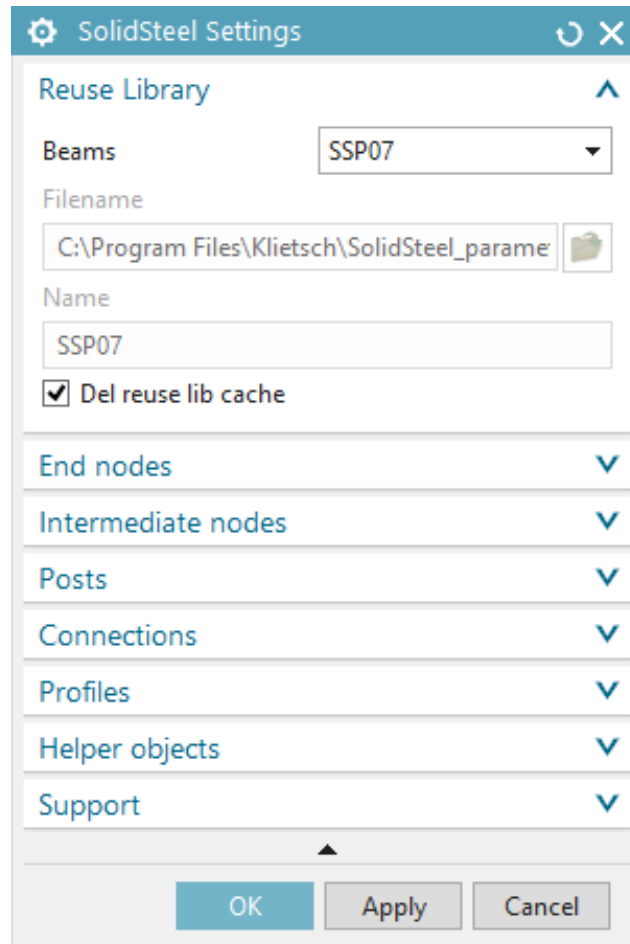


Update dialogue of SolidSteel parametric

After clicking Apply or OK, the profile length, the angle of the connections and things like that will be updated by the SolidSteel parametric kernel.

Settings

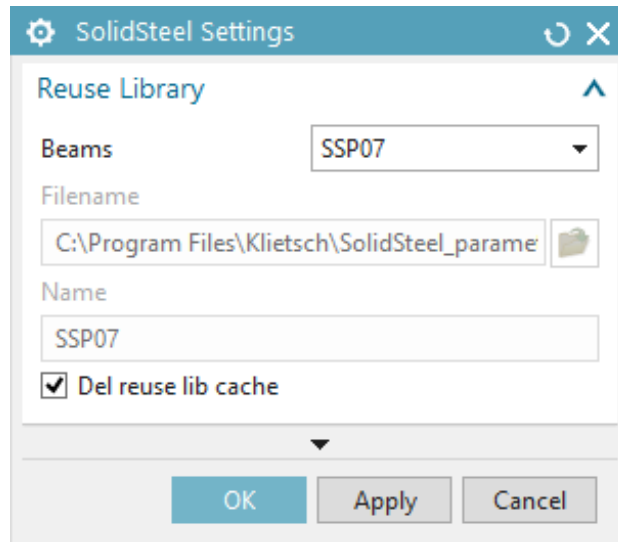
Some settings and preferences can be set in SolidSteel parametric for Siemens NX. You can do that by opening the „settings“ dialogue in the SolidSteel ribbon bar.



Settings dialogue of SolidSteel parametric

Reuse Library

The most important settings are those for the Reuse Library.



Reuse Library settings

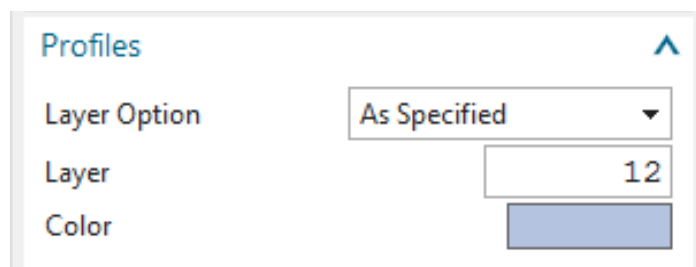
If there are any issues with the profile dialogue:

Make sure that the path to the profile data is correct!

First select the version for the current installation and check if the path is correct. Then type in a name. Important is to delete the Reuse Library cache by clicking the check box „Del reuse lib cache“. Then click apply. The process may take some minutes.

General Settings

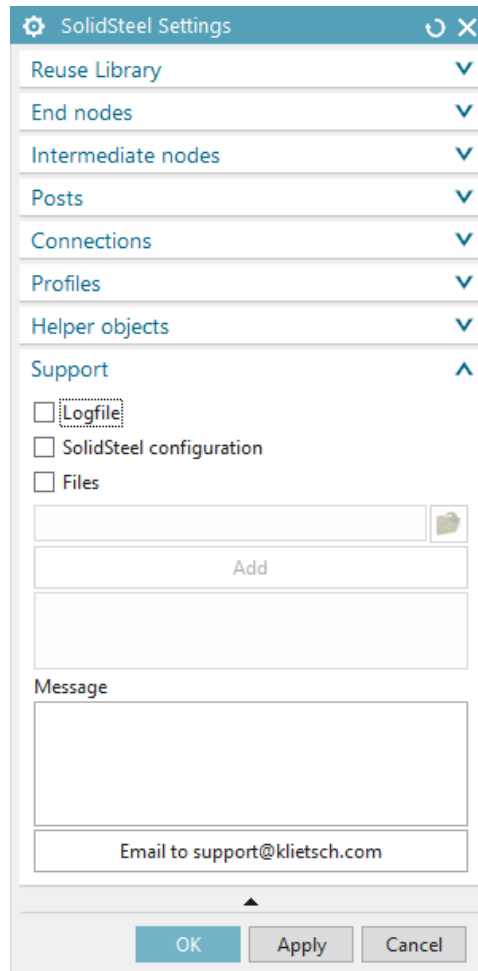
In the SolidSteel settings dialogue some general settings like changing the color of the profiles are possible. To do that open the dialogue and change the colors the way you like it.



Changing the color of the profiles

Support eMail

If there are any issues with SolidSteel parametric for Siemens NX you can send us a support mail from the settings dialogue.



Support eMail from the settings dialogue

Please check the boxes for the „LogFile“ and your „SolidSteel configuration“. This will help a lot. Of course you can add additional files like your current construction or things like that. Please describe the error as accurately as possible.

Now click „Email to support@klietsch.com“. Then an Email will be sent by Outlook. If you have no Microsoft Outlook, please send the eMail to support@klietsch.com manually.