SolidSteel
for creo elements/direct

news in update 7.0.6

DSTV - NC-Data news in Update V7.0.6

In addition to general improvements, the functionality of SolidSteel has been greatly expanded with the update 7.0.6, especially in the area of NC data. In this document you will find detailed information on how to use the new functions.

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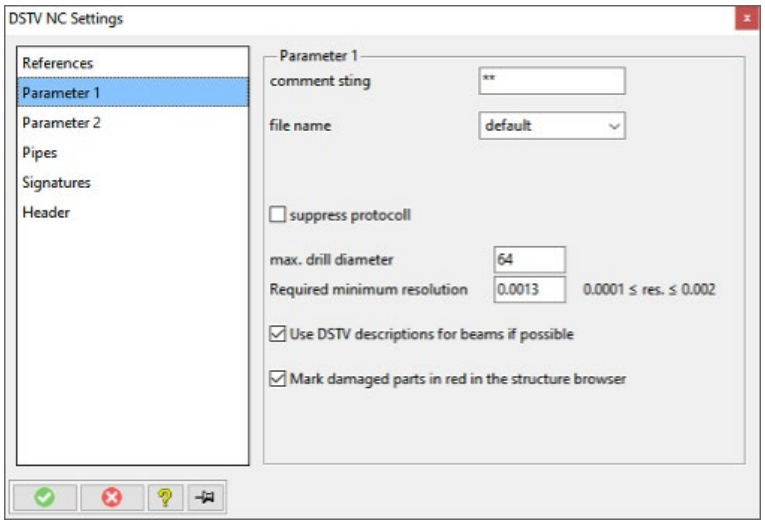
Settings



DSTV NC Settings

Beam descriptions

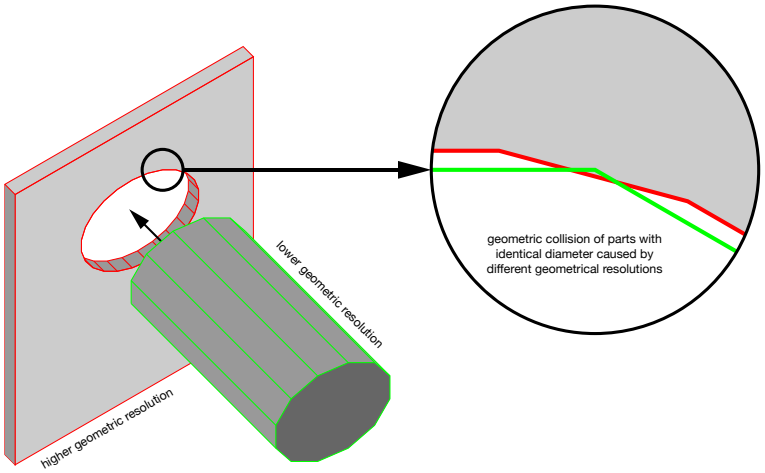
Up to now the descriptions were taken from the internal SolidSteel data. Now the Descriptions according to the DSTV documentation can be used if available for the beam.

Part checking

Corrupt parts (incomplete faces, loose contours, wrong face normals etc.) lead often to problems (program crash). To prevent this we now call a part check before scanning and creating NC. If configured these parts can be displayed in red color in the structure. Furthermore no NC data will be created for those parts.

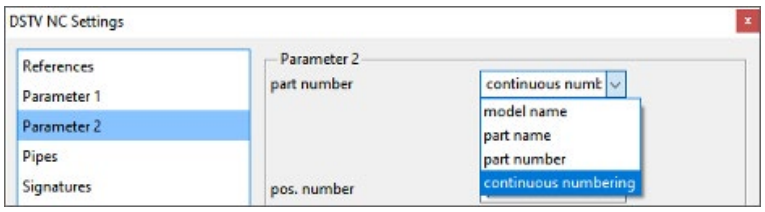
Geometric resolution

When scanning or recognizing beams a minimum geometric resolution of the parts is required to operate properly. In practice a minimum value of 10^{-3} turns out to be as a good compromise. Depending on the data source it is now possible to set the minimum resolution to a value in the interval from 10^{-4} to $2*10^{-3}$.



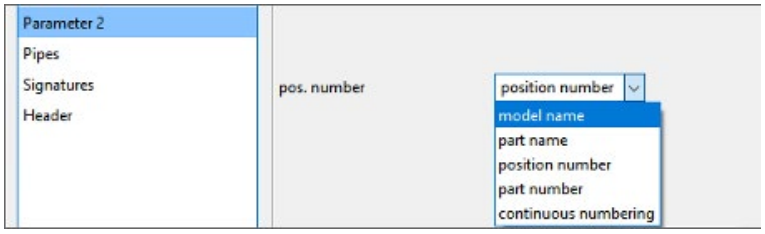
Example for different geometrical resolutions.

Part number



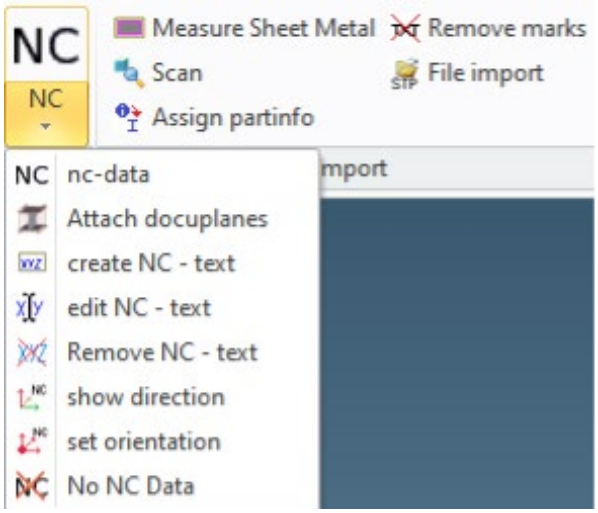
Part and model name can now be used as part number.

Position number



Part and model number can now also be used for the position number.

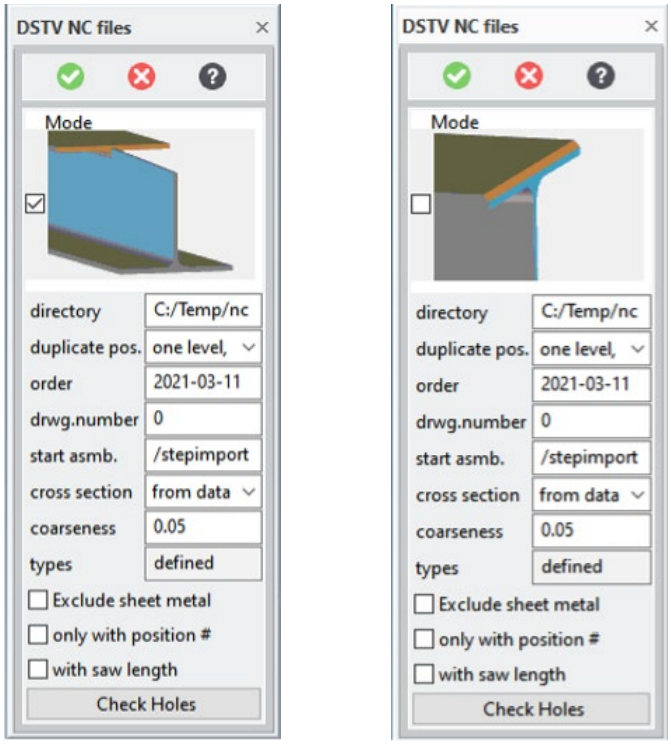
Menu



The menu with the functions for recognizing beams and creating NC data was reorganized to allow a better work flow. Functions that a used more often are now directly accessible.

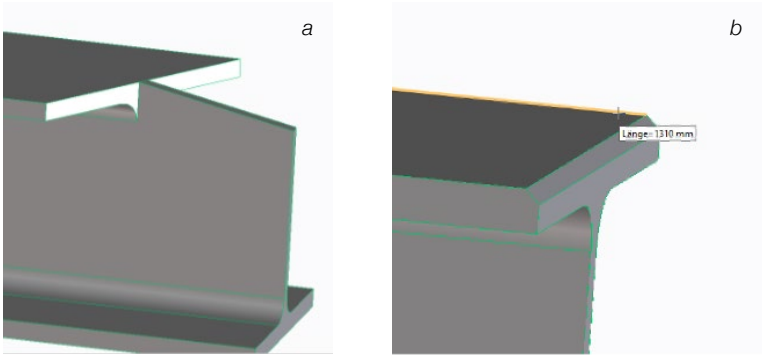
New and modified functions

NC data generation



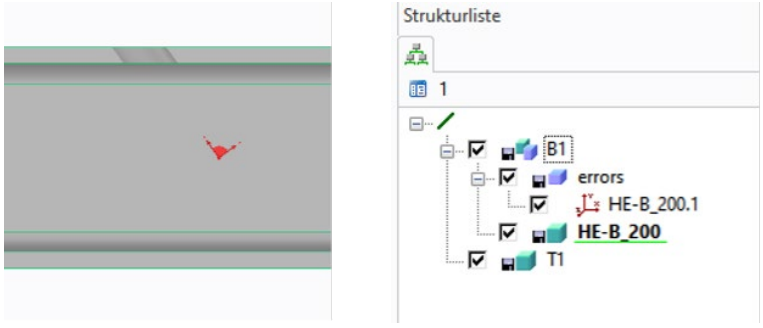
Dialog NC data generation

NC data of beams with cuts as shown in the left picture where a remaining part of the web coincides with the flange could not be created correctly. To create NC Data from those beams we added a new analyze mode. For beams with a weld fase the normal mode must be used.



Examples for web at flange (a) and fase (b)

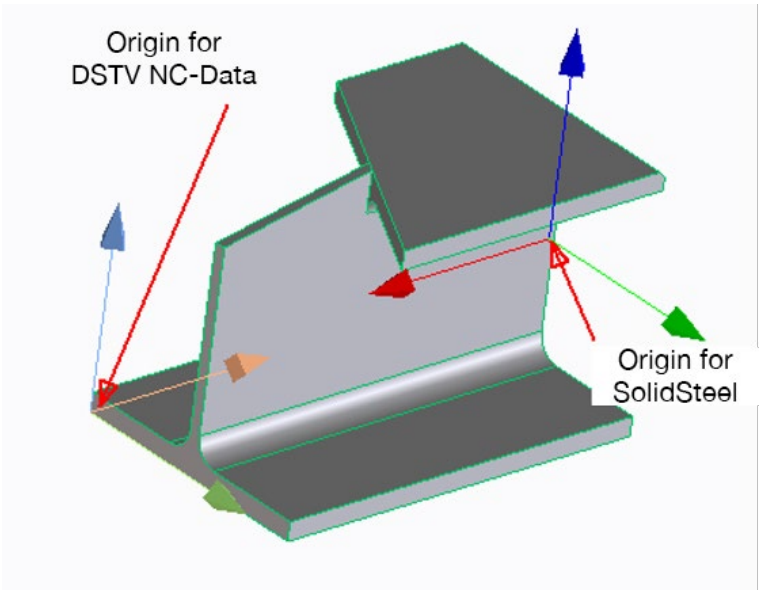
The button |Check holes| checks for through holes with an axis that is not orthogonal to the surface. If the check finds holes like this, you will get an alert and a container errors is created below the start assembly with coordinate systems that mark the position and direction of any of the erroneous holes. The w – axis shows the direction of the axis of the bore hole.



Beam with non ortogonal hole and structure list with reported error

Dialog show direction

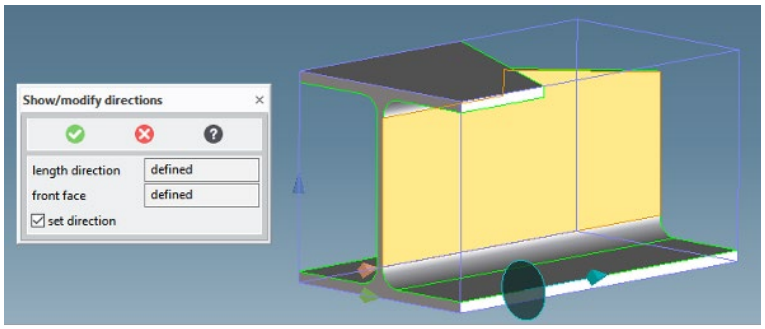
With this dialog it is possible to visualize the positions and directions of the DSTV NC and internal SolidSteel coordinate systems. Furthermore the size of the bounding box is displayed.



The "paler" coordinate system on the left marks the origin of the DSTV NC system. The green arrow shows the direction of the front view, and the blue arrow shows the up - direction.

Dialog set orientation

Sometimes the orientation of the beam is not recognized correctly during the scan process. Also one might determine manually the directions of symmetric beams. With this dialog the directions can be determined by means of the extrusion direction and a face with a normal in front direction. After selecting direction and face the new orientation is displayed with a coordinate system. To use the new direction the checkbox ☐ set direction must be checked.

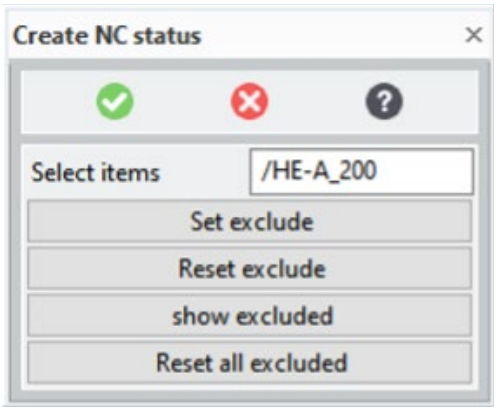


Dialog set orientation

Dialog No NC data

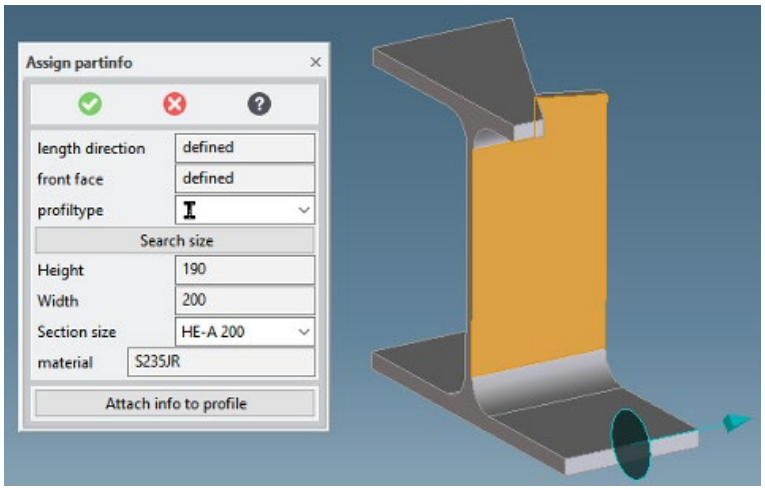
When a construction contains parts that should be excluded from NC data generation this dialog can be used to attach an information for the NC Data program to skip them.

One or more parts or assemblies can be selected. If an assembly is selected all parts below that assembly are excluded. To reset the exclude information from the selected objects the |Reset exclude| button can be used. |Show excluded| modifies the display list so that only excluded objects are shown. Reset all excluded restores the display list to its prior state.

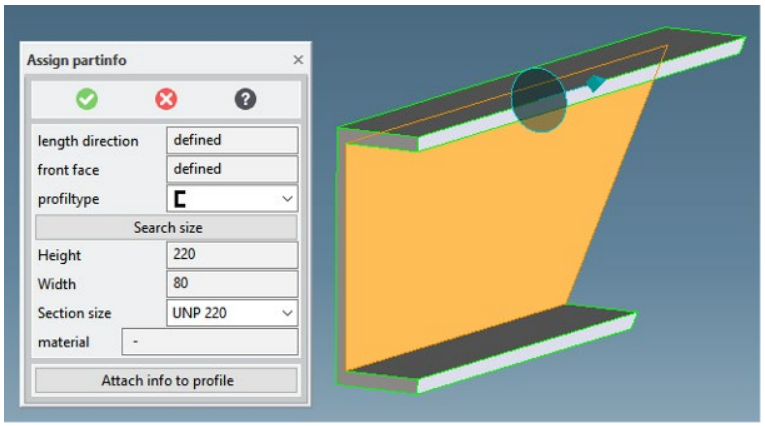


Dialog assign partinfo

Beams that cannot be recognized because of poor geometric resolution or are too short or have modifications that prevent a successful scan, can be assigned a beam information and direction manually. Often this allows to get the NC data. Of course the user must take care to enter values that makes sense to get a correct result. After assigning an extrusion direction and selecting a face with normal direction to the front side and a beam type the dialog can try to find beams that fits the size with the |Search size| button. If the search is successful the selection size range contains all matches. Furthermore a material can be attached.



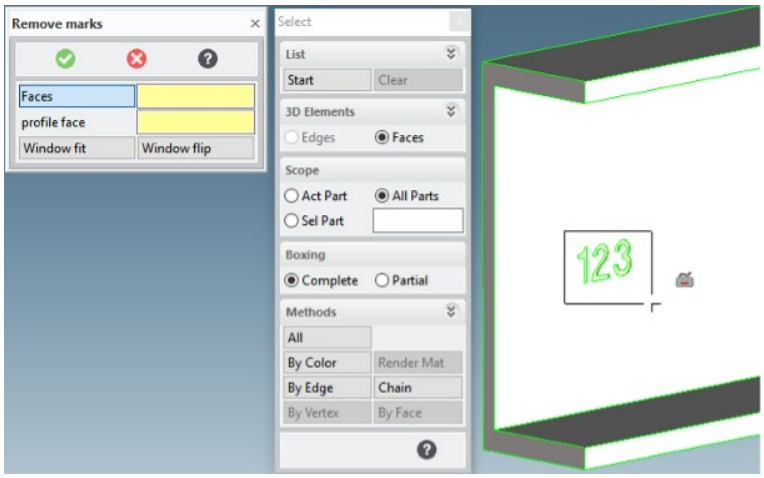
The example shows a beam that is not recognized during the scan. After selecting extrusion direction, a front face and a beam type the correct beam is found and NC data can be created.



This also applies for profiles with missing chamfers and bevels as shown in the example.

Dialog remove marks

Occasionally it happens that the part supplier has stamped information into the profiles. The NC data program would try to recognize the imprint as machining operations, but this fails and therefore it must be removed.

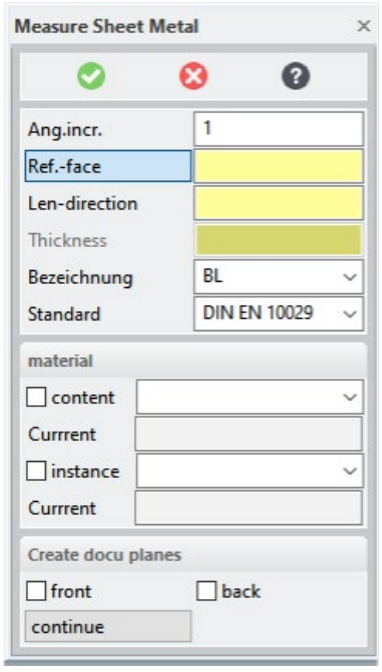


After the user has drawn a frame around the indentations and selected a reference surface, the imprint is removed.

Important:
This only works for imprints, not for reliefs!

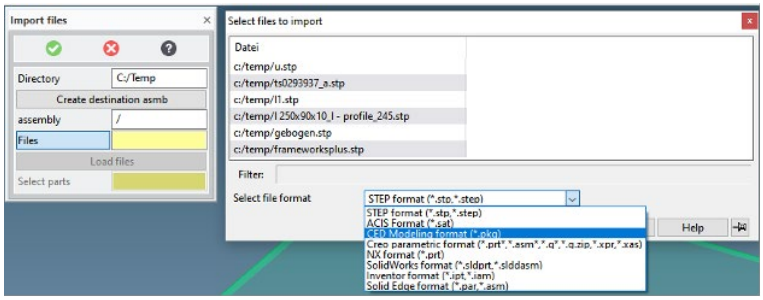
Dialog Measure Sheet Metal

This dialog was expanded by an |continue| button, so the user can remain in the dialog after collecting data of one plate. Furthermore the last part is removed from the display list.



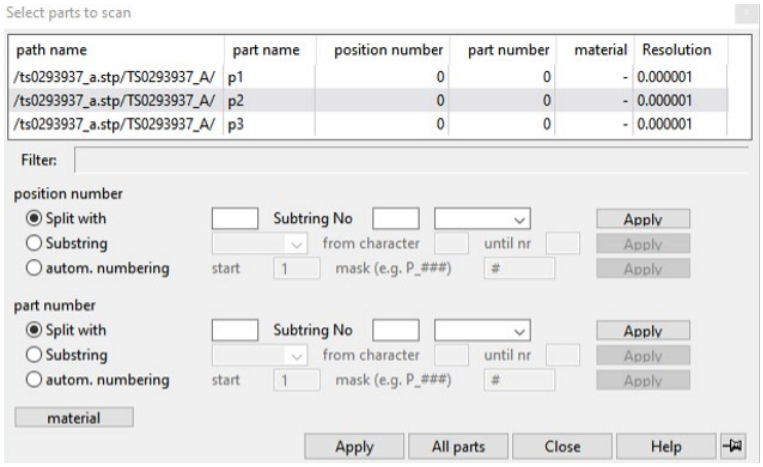
Dialog File Import

This dialog combines some steps to speed up the process of importing and scanning beams.



After selecting the import directory all files with the file format chosen will be listed. A destination directory can be selected or if desired created. Hitting the button |Load files| *will load all selected files in the destination directory. Subsequently all parts loaded will be shown in a table.*

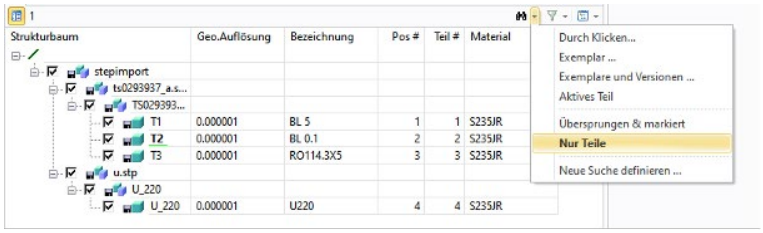
The parts selected in this table will be scanned. Also part numbers, position numbers and materials can be assigned to the parts. This works for all displayed parts in the table. Double clicking on an entry opens a dialog so one part can get individual data.



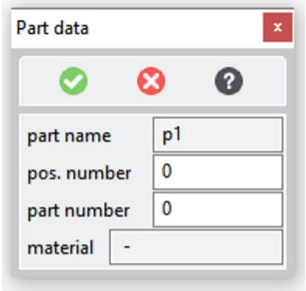
Split with *uses a pattern to split path name or file name and uses the nth substring.*
Substring *uses a part of the path name or filename defined by start and end character.*
autom. Numbering *just creates a string according to the mask, where # is replaced by the number.*

The number is increased after each part and is left filled with zeroes according to the number of # characters, e.g. P_### is translated to P_001, P_002, P_003...

After selecting the desired parts, clicking on Apply and hitting the scan process will be started and the result is displayed thereafter. Additionally a browser view is activated that shows geom. resolution, description, position number, part number and material. Also a browser search is added that expands the assemblies to show the parts.



Right click on a part in the structure browser with pressed |CTRL| *key starts a dialog to edit pos. number, part number and material.*



Further Information

Our apps for creo elements/direct include many tools that make your daily work faster and easier while mapping the complete process chain in steel construction. Visit our website for more information.

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