

Hotline-Support KOSY/MCS

Category 2: **Software**

Error: **DXF-Import not correct/fragmentary**

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You draw in your regular CAD-program and want to continue by machining the parts via *nccad*. On the way from drawing to machining you need 3 function modules resp. 3 different programs:

- **CAD** (Computer Aided Design)
Drawing in a 2D- or 3D CAD-program
- **CAM** (Computer Aided Manufacturing)
Define the machining parameters, i.e. Speed, infeed, tool etc.
- **CNC** (Computerized Numerical Control)
Numerical machine control supported by computer.

nccad contains all 3 function modules for fast workshop oriented programming WOP. However, when you use 3 different programs, because *nccad* doesn't meet all your demands, an export takes place from one program to the other and subsequently an import, therefore there are **2 interfaces**:

- **From CAD to CAM**
For **2D-drawings** the **DXF-format** is used. The drawing shows the contours in one view (top view). Inside of CAM it becomes a 2 1/2-D machining, i.e. The Z-axis goes inside, the X- and Y-axes follow the contour.
- **From CAM to CNC**
A NC-program is generated in a CAM-program, it must correspond to the "dialect" of the machine control. The adaptation is made in the so-called **Postprocessor**, which must be available as part of CAM related to the used machine.

At each interface the compatibility of the systems must be guaranteed, which is not always the case. It starts with the question of responsibility, who is responsible that the systems match and who helps if not. An example: Do we adapt the importfilter inside *nccad*, or can the export from the CAD program be adapted? We can help in many cases, but please keep in mind that there are hundreds of CAD-programs and special cases, which make it difficult.

In the case treated here, a drawing in 2D DXF-format is imported in *nccad*, then CAM and CNC inside *nccad* is used. Only 1 export/import is necessary:

1. General Information about DXF-Import

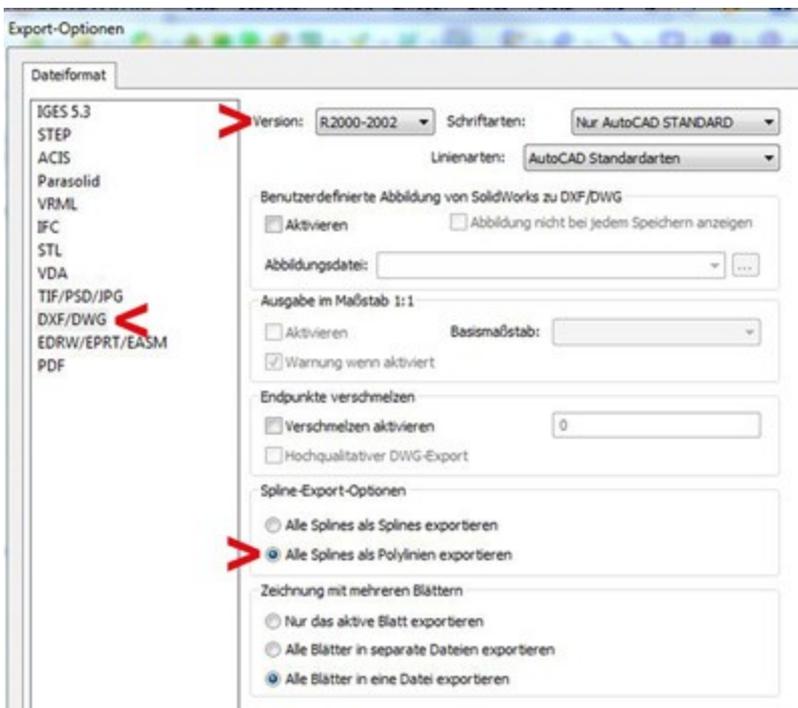
The DXF-format comes from the world of "AutoCAD" and is widespread, but has not remained constant in the course of the years and **it is not standardized**. Therefore it is not very compatible. We can help in many cases when you send us the DXF-files making problems. But it is not always possible to adapt the importfilter.

In most cases the following tips for **parameter settings and proceeding for DXF-export are helpful**:

-  Export **2D-drawings** in the corresponding view / level (topview for the machining you want to do).
- Take into account the later import in *nccad* in the software you are drawing with, it simplifies later work:
 - position of the zeropoint according to clamping-in position
 - positive X- and Y-coordinates (draw in upper right hand side)
 - closed contours (no gaps or overlappings with contours).
- Export on the base of an older version of AutoCAD, e.g.. R14. With higher versions there is the risk that not all parts are imported in original way. Try with the lowest version when exporting or make some experiments.
- At the end of constructing take drawing parts "back to origin" (function in some CAD-programs).
- Export **no splines** (special curves), but change the curves into polygons resp. export them as polyline. The reason: the calculation formula which is used in CAD for curves, is not known to *nccad*.

1.1 Export Settings

All CAD-programs can be set to DXF-export. In the case of SOLIDWORKS, for example, after clicking **Save as** and after selecting DXF-format you find in the export window the button **Options**, leading to the window **Export-Options**:



Export-Options in SOLIDWORKS

For *nccad* we recommend:

- Version: R14 to R2000
- Spline export: As Polylines

Other settings must be made individually, e.g.:

- Used defined representation (export not all contours/parts)
- Scale
- Join endpoints (generate closed contours)
- Export the active sheet only or all sheets

We recommend to make a test with your special demands.

It is only a few clicks to do and it seems normal that things have to match.

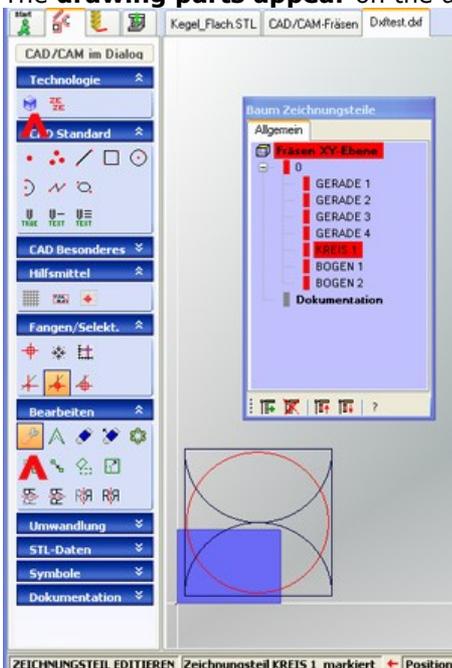
When you get DXF-files from partners who do not deal with details, a format converter can be helpful; it is available in different levels, even for free. Another possibility is the exchange of files in 3D-format (Step, DWG, Iges..), if you have got 3D CAD. The advantage is that you you'll get an overview on the whole construction. Conversion as well as DXF-export are then made by your CAD-program.

The target is always the machine and machining, but the CAD-program is not able to set the parameters for machining nor can it control it.

1.2 Import of DXF-files by *nccad9x*

The **import** in *nccad* requires the following **operational steps**:

1. Call up DXF-import in the menu **File/Import** or in the icon bar under **Import/DXF**, select the file and open it. As example from the folder `C:\User\Public\MAX\nccad\Mill\Beispiele...\DXF` the file `DxfTest.DXF`.
2. The **drawing parts appear** on the drawing area and in the tree **Drawings**.



It may happen that the following **particularities** arise:

- **Drawing elements** are partly or as a whole **in negative**: The white crosshairs for the workpiece zeropoint shows automatically. In order to "take it out of the corner" use the function **Shift view** in the operator bar and the function **SHIFT** in der icon group **Edit**.
 - **Drawing elements are outside** of the workpiece representation (see picture above): Select the function **WORKPIECE** in the icon group **Technology** and change the measures.
3. **Re-work the drawing elements** (redraw parts or delete them etc).
 4. If necessary: **Generate new groups** and drag and drop the parts into the groups for a certain machining.
 5. Change **order of groups** by drag & drop, so that it corresponds to the order of machining.
 6. **Apply technology (machining parameters) to the drawing elements** via the function **EDIT DRAWING ELEMENT** in order to prepare them for machining.
 7. Start simulation and maybe go back to drawing functions for correction.

We recommend to make some tests first: draw simple parts in the CAD-software and export them as DXF-file, import them in **nccad** and examine them.

1.3 Settings of importfilter

The settings are preset. After import all drawing elements appear in group 0 in black.

Send us the DXF-file which causes problems . We can help in most cases.