

DXF Export Filter (Module EXPDXF)

General

When exporting into a DXF file, some information gets lost. This is caused by several limitations in the DXF file format.



Definition of line patterns, layers and blocks are retained. Though, due to the fact that names within a DXF file may only consist of very few characters, the names of these elements have to be preceded by continuous numbers, so that they remain definite. Line patterns get the prefixes L01 to L50, layers the prefixes E001 to E500, and blocks the prefixes B00001 to B99999.

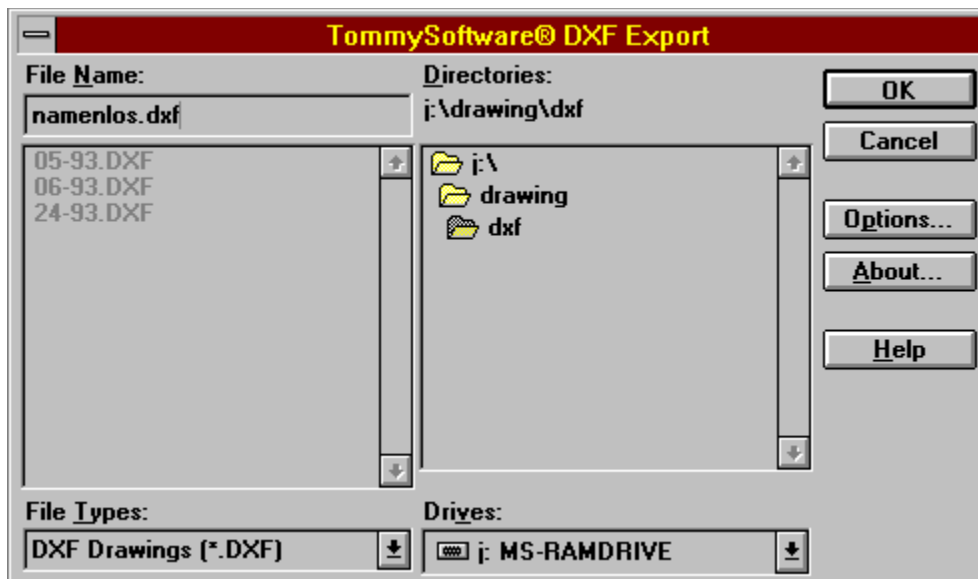
External references to symbols are only supported if the option "Maintain Block Structure" is not enabled.

As line patterns in DXF files can not be depending on the line width, those line patterns are calculated for a line width of 0.5 mm.



Fillings are ignored during conversion. If an object was only filled, but not framed, it will be framed in the file, though, to be visible.

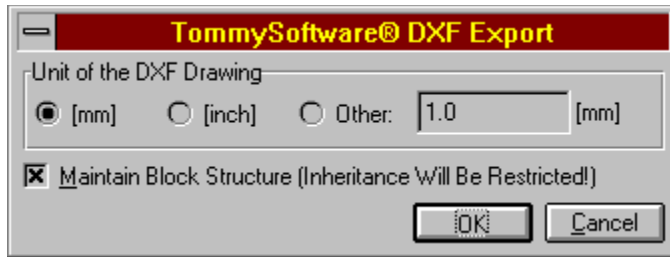
File Selection

To access an explanation of an individual element in the dialog box shown below, click the left mouse button on the element. Explanations are available whenever the mouse pointer changes from an arrow () to a hand ().



Options

To access an explanation of an individual element in the dialog box shown below, click the left mouse button on the element. Explanations are available whenever the mouse pointer changes from an arrow () to a hand ().



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Clicking on this button will close the dialog accepting all changes. Any changes or operations specified will be carried out.

Clicking on this button will close the dialog, without accepting any changes. Any following operation will not be carried out.

This text shows the path of the directory whose contents are shown in the file list. This path is always shown in front of the file name in the "Name" field.

The name of the file to be used is entered in this field. If there is a file in the list whose name begins with the character entered, this file is put at the top of the list.

The extension for the files to be displayed can be selected from this list. There are usually only two or three file extensions available. If a standard ending (DXF) is chosen, it is forced. This means that the chosen ending is applied to all files.

For example, if a drawing file called `MYFILE` is entered, the extension DXF will be used, the resulting file will be called MYFILE.DXF.

All the files in the current directory with the specified extension are shown in this list. Clicking once on a filename puts that name in the "Name" field, where it can be changed. Double-clicking on a file name selects that file and closes the dialog as if the "OK" button had been clicked on

All the sub-directories in the current directory are shown in this list. Double-clicking on a directory name will open or close that directory.

All available disk drives are shown in this list. Selecting a disk drive name makes that drive the active drive, and the path shows current root directory of that drive.

Clicking on this button displays brief information about this filter.

Clicking on this button allows to edit the options for the current file operation.

Clicking on this button display the help text for this dialog window.

Clicking on this button accepts the current file name. If a standard ending has been chosen from the list of extensions (DXF) this is forced. This means that the file will use the selected extension, whichever extension is entered.

The program checks whether the file already exists. If the file to be loaded does not exist, then an error message appears and the dialog remains open. If a file with the same name as the one to be saved already exists, a message appears and you can choose whether or not to overwrite the existing file.

Clicking on this button will close the dialog, without loading or saving a file.

If this radio box is selected, the DXF drawing will be created based on Millimeter unit.

If this radio box is selected, the DXF drawing will be created based on Inch unit.

If this radio box is selected, the DXF drawing will be created based on the unit given in the edit box in millimeters.

Enter here the custom unit in millimeters that the DXF drawing shall be based on.

If this check box is enabled, the structure of blocks, groups and instances will maintained during export. Since the DXF format uses another method of inheritance than TVG 4.0 does, some problems may occur. To avoid those, you have to clear the check box. All blocks and group will then be resolved during export. If a drawing contains external references, this check box must be cleared for export.

