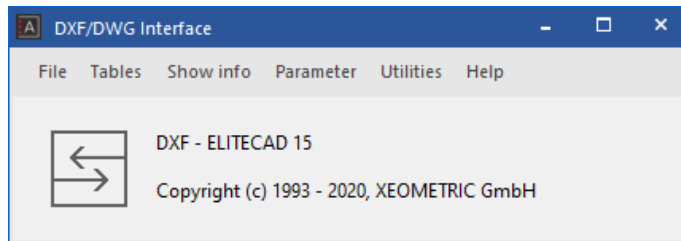


COURSE UNIT



DXF - Data Exchange

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INTRODUCTION

What is DXF?

DXF stands for "Drawing Interchange Format" or also "Drawing Exchange File". DXF is a standard industry format, not a norm as such, and is subject to continuous development.

DXF is a standard format for data exchange that can be written and read by all common CAD systems.

Set-up of a DXF file

The DXF file is divided into five sections:

HEADER

General description of the DXF file, DXF version, system settings, etc.

Important: Plan format, dimension unit and scale are not described in the DXF file!

TABLES

In this section, tables or lists with layers, stroke types, character types, dimension types, etc. are defined that are used in the sections BLOCKS and ENTITIES.

BLOCKS

This part of the file contains drawing elements that are combined into groups, e.g. walls, windows or doors that can be inserted into a drawing as a whole.

ENTITIES (basic graphical objects)

This section contains all the drawing elements of a CAD drawing: lines, circles, arcs, etc. or references to inserted element groups that were defined in the BLOCKS section.

EOF (end of file)

Example: DXF file

The file is in ASCII format and may contain several thousand lines.

0	10	7
SECTION	0.0	Standard
2	20	9
HEADER	0.0	\$CLAYER	91
9	9	8	1
\$ACADVER	\$LIMMAX	0	62
1	10	9	0
AC1021	12.0	\$CELTYPE	92
9	20	6	-2
\$ACADMAINTVER	9.0	ByLayer	340
70	9	9	0
25	\$ORTHOMODE	\$CECOLOR	93
9	70	62	0
\$DWGCODEPAGE	0	256	40
3	9	9	0.045
ANSI_1252	\$REGENMODE	\$CELTSCALE	309
9	70	40	GRIDFORMAT_END
\$LASTSAVEDBY	1	1.0	95
1	9	9	32
jenny	\$FILLMODE	\$DISPSILH	302
9	70	70	GRIDFORMAT
\$INSBASE	1	0	1
10	9	9	GRIDFORMAT_BEGIN
0.0	\$QTEXTMODE	\$DIMSCALE	90
20	70	40	1
0.0	0	1.0	-2
30	9	9	340
0.0	\$MIRRTEXT	\$DIMASZ	0
9	70	40	40
\$EXTMIN	0	0.18	0.045
10	9	9	309
28.83155830327638	\$LTSCALE	\$DIMEXO	GRIDFORMAT_END
20	40	40	309
17.84509184718143	1.0	0.0625	TABLEFORMAT_END
30	9	9	1
0.0	\$ATTMODE	\$DIMDLI	CELLSTYLE_BEGIN
9	70	40	90
\$EXTMAX	1	0.38	3
10	9	9	91
45.84244948693004	\$TEXTSIZE	\$DIMRND	2
20	40	40	300
34.90250098088643	0.2	0.0	_DATA
30	9	9	309
0.0	\$TRACEWID	\$DIMDLE	CELLSTYLE_END
9	40	40	0
\$LIMMIN	0.05	0.0	ENDSEC
0.0	9	9	0
0.0	\$TEXTSTYLE	\$DIMEXE	EOF

PRELIMINARY INFORMATION

Order and Problem

Today there is an increased need to accept CAD data directly from project partners and therefore to save the time and effort required to enter it a second time.

All too often however, CAD data exchange is inefficient and error-prone. Far too much quality, time and money are still being lost.

Technology and Organisation

What are the reasons for the most frequent data exchange problems? The technology is seldom the problem. Today's conversion programs are generally well-engineered and good quality.

For successful data exchange, in addition to technology organisation is the major factor.

Without a minimum of organisational measures to accompany the project, problems are inevitable. Usually the necessary basic knowledge is not present or particularly straightforward; organisational tools are missing.

Difficulties occur repeatedly for data in DWG format:

AutoCAD plans with referencing objects, model and paper format are not very suitable for data exchange. AutoCAD users must also prepare data transfer. Converting to DXF format can usually resolve these problems.

Protocol (delivery note), data organisation print-out or PDF file

The protocol (delivery note) makes every DXF data exchange traceable. The data receiver then receives all information required for secure data transfer (example: page 5).

The data organisation shows the most important display standards and system settings. On a different CAD the relevant settings can then be predefined and processing continued in the same structure (see example on page 6).

A hard copy printout (e.g. a PDF file) must always be included with the exchange. Your partner can then immediately establish whether the transfer is truly complete.

Test drawing

Presettings can be determined in advance and adjusted with a simple test drawing (see example on page 7).

The State Building Department of the Zurich canton requires a test with additional work (see example on page 8).

IMPORTANT:

Do not wait until the last minute to carry out a DXF data exchange. A test run at an early stage can save you a lot of trouble.

Find out exactly what data your partner requires. Excess data only makes the exchange more difficult: transfer only as much data as you require.



















PROTOCOL (DELIVERY NOTE)

Protocol	Delivery note			
Communication partner - Data producer - Data receiver				
Project				
Name of plan	Inspection plan	Unit	Format	Scale
Systems - Operating system - CAD system - DXF version - Data organisation				
Data carrier - Operating system - Size - Backup format - Compression				
Delivery conditions - Agreements - Rights - Costs				
Miscellaneous				
Place, Date	Signature			

DATA ORGANISATION

Data organisation	
Project management	
Object breakdown	
Display standards <ul style="list-style-type: none">- Coordinates system- Unit of measure- Pen colours- Line types- Character set- Special characters- Fonts- Dimensions- Hatch- Infill hatch / colours- Layer names- Block names	
Miscellaneous	
Place, Date	Signature

TEST DRAWING

TESTZEICHNUNG	DXF - DATENAUSTAUSCH
	FORMAT A4
SONDERZEICHEN	EINHEIT CM
ÄÖÜ äöü °±∅	MST 1:1
STIFTE	
	WEISS 0.70
	ROT 0.50
	GELB 0.35
	GRÜN 0.25
	CYAN 0.18
	MAGENTA 0.13
	BRAUN 0.35
	OLIVE 0.35
	BLAU 0.35
LINIEN	
	CONTINUOUS
	HIDDEN / VERDECKT
	DOT / PUNKT
	DASHED / STRICHLINIE
	DIVIDE / GETRENNT
	DASHDOT / STRICHPUNKT
	BORDER / RAND
	CENTER / MITTE
	PHANTOM

TEST DRAWING HBA CANTON ZH

Testzeichnung		zum Datenaustausch		18.09.10	
Empfänger:	Hochbauamt Kanton ZH	Absender:	Roland Messerli AG Informatik	Einheit: / Format:	cm / A3
Adresse:		Adresse:	Pfadackerstrasse 6	Massstab:	1:1
PLZ, Ort:		PLZ, Ort:	8957 Spreitenbach	CAD-Software:	EliteCAD V12
Tel.:		Tel.:	056 418 38 00	Betriebssystem:	Windows7
E-Mail:		E-Mail:	cad@messler.ch	DXF-Version:	AC-Rel.2010

N	Color	(mm)	Line	Text	Linetypes	Bsp_Line	Textwerte (H/B/W)
1	white	0.70		Test	continuous	1	2.5 / 1.25 / 0
2	red	0.50		Test	verdeckt	2	ABCDEFGHIJKLMNPOQRSTUVWXYZ
3	yellow	0.35		Test	punkt	3	1234567890
4	green	0.25		Test	strichlinie	4	
5	cyan	0.18		Test	getrennt	5	
6	magenta	0.13		Test	strichpunkt	6	
7	braun	0.35		Test	rand	7	
8	olive	0.35		Test	mitte	8	
9	blau	0.35		Test	phantom	9	

ASCII-Tabelle CAD400																									
0	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16	17	18	19	20	21	22	23	24	25
0	(2	<	F	P	Z	d	n	x												Ø				
1)	3	=	G	Q		e	o	y																
2	*	4	>	H	R	\	f	p	z												ß				
3	!	+	5	?	I	S]g	q	{																
4	"	,	6	@	J	T	X	h	r												ä	ø			±
5	#	-	7	A	K	U	_	i	s	}															
6	\$.	8	B	L	V	^	j	t	~											ö	Ä			
7	%	/	9	C	M	W	a	k	u												£	ü			
8	&	0	:	D	N	X	b	i	v													Ö			
9	'	1	;	E	O	Y	c	m	w												°	§	ı	Ü	

Masslinien und Koten	

Testelemente					
Schraffur BN	Musterschraffur	Filling rot	Polygonzug	Polygonfläche	Kreis
Bogen		3D-Extrude	3D-Fläche	3D_Würfel	3D-Wand
					A1—E-
(Punkte=Text)	Ellipse	Tang.Bogen	Tangenten	7-Eck	A1E41—E-
					A1M16—E-
					A1Z21—T-
					Layer-Test

COMMENTS

Protocol (delivery note)

A protocol (delivery note) is an integral part of every DXF data exchange. The unit, format and scale must always be included in the delivery note. These values are not written to the DXF file and are therefore not taken into account during import. These preliminary settings then mean there is no need for adjustment to the correct size.

Data organisation

Check the data organisation for every new request. Some display standards (data organisation) can also be checked with the test drawing.

Test drawings

It is important that the drawings are created on a separate CAD. Test drawings already imported via DXF lead to falsified results.

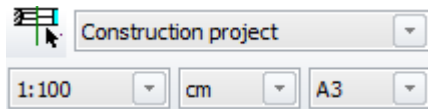
The test drawings accompany this course unit in the ELITECAD format or can be obtained via the Hotline.

DXF - IMPORT FILE

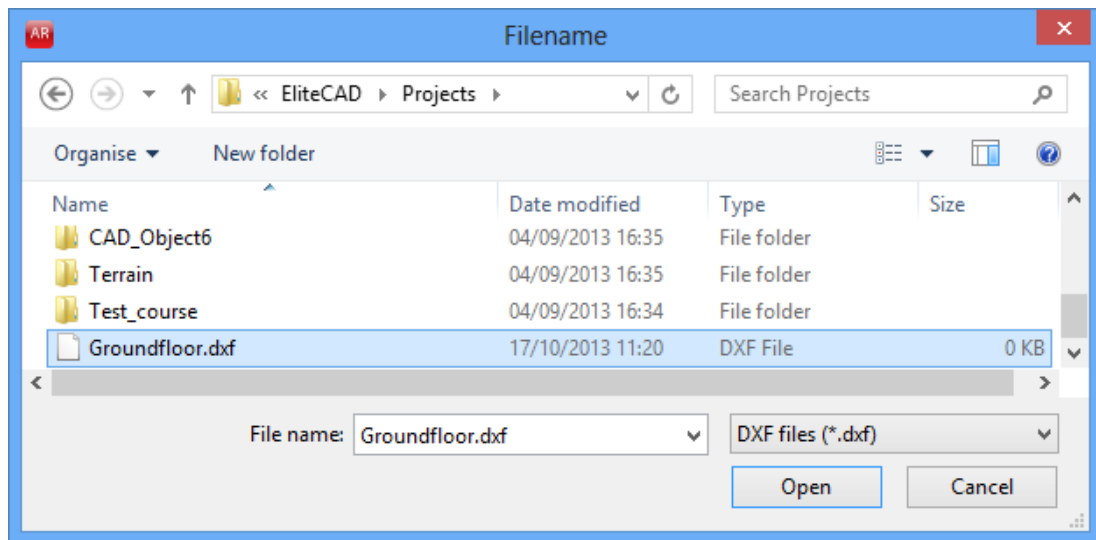
Spontaneous users

Protocol (delivery note) should be present.

1. Set the unit, format and scale!



2. Import the DXF file via the ELITECAD main menu
File > Open ...



3. Search, select and open DXF file

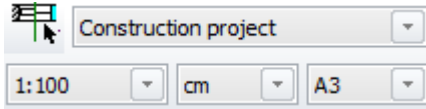
Comment:

The standard configuration "Dxf.dxd" is always used for this procedure.

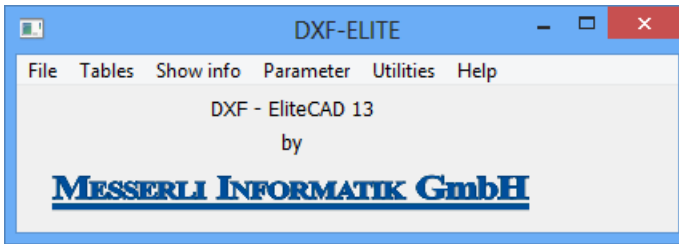
DXF menu

Protocol (delivery note) should be present.

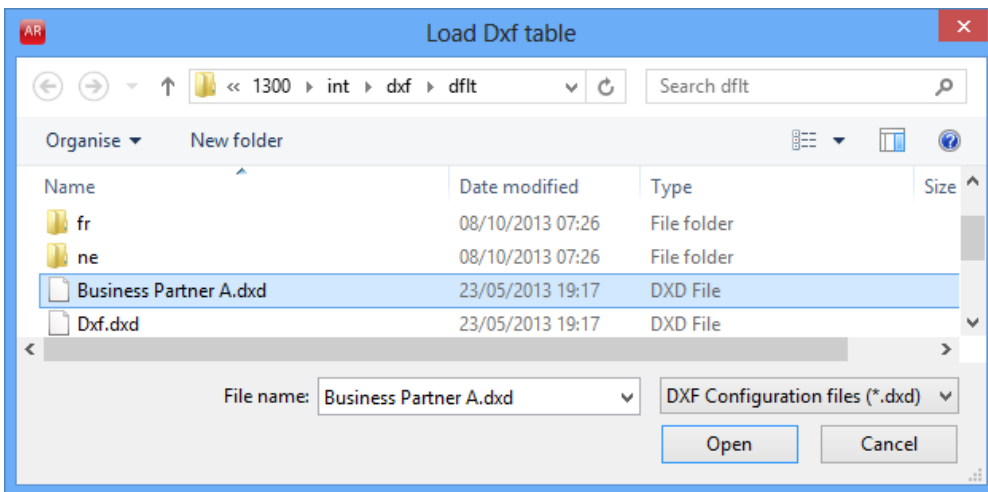
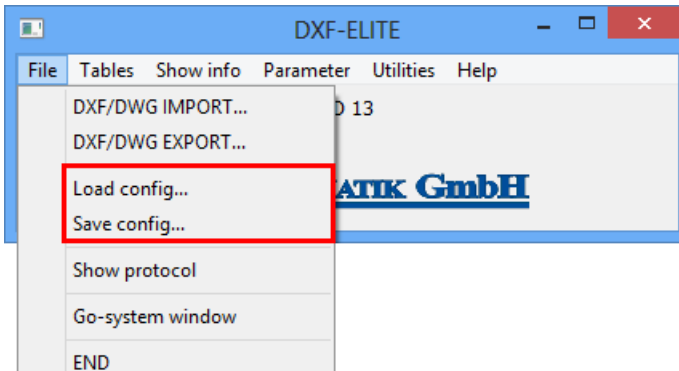
1. Set the unit, format and scale!



2. Open the DXF menu in the ELITECAD main menu
File > Interfaces > DXF/DWG



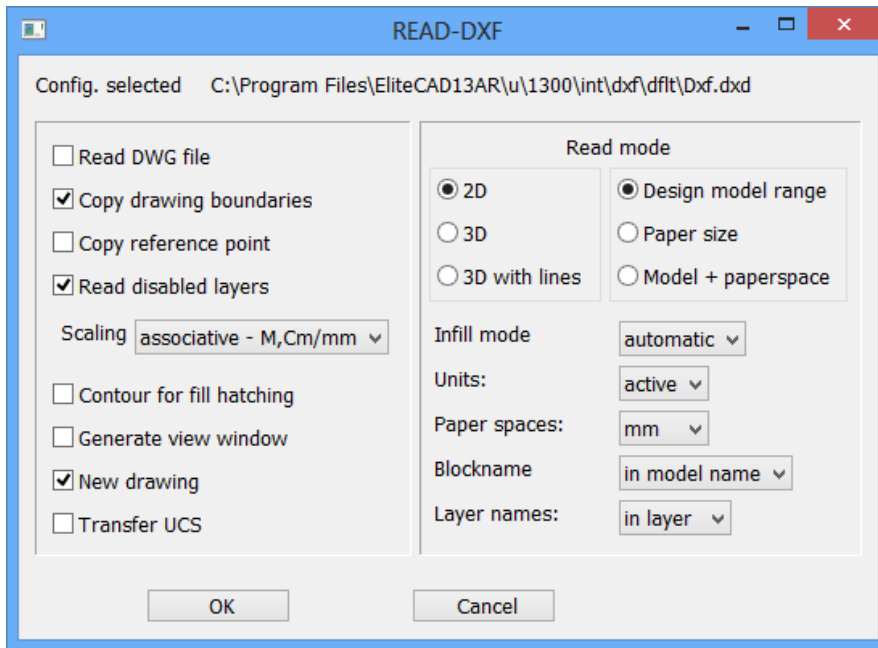
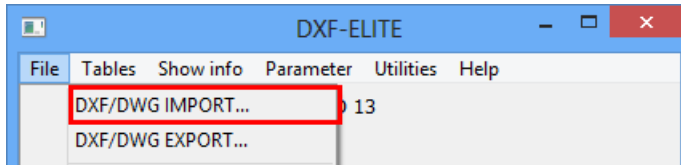
3. Import configuration
DXF - ELITE: File > Load configuration...



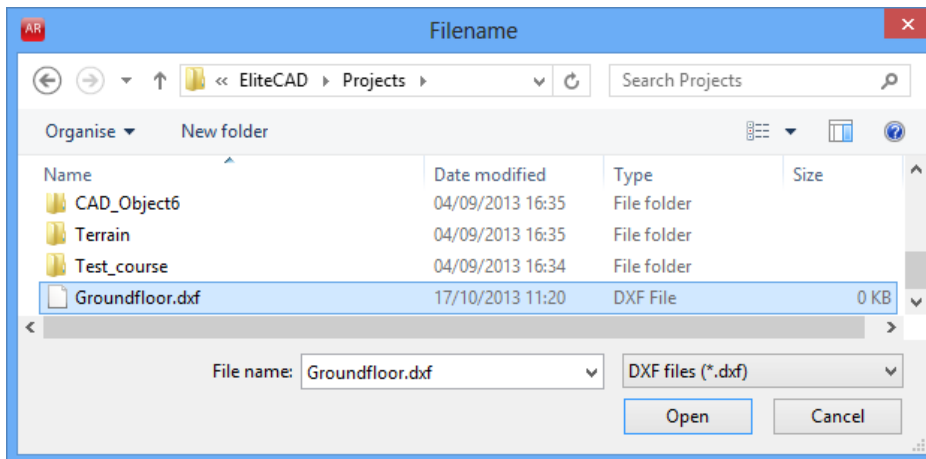
4. Select and open the desired configuration

If necessary, the tables ([pages 32-37](#)) can now be adjusted with the help of later declarations.

5. Import DXF file



6. Check parameters, adjust if necessary, and then click OK



7. Search, select and open DXF file

8. Check the imported plan and info window.

Interpret messages, correct procedure if necessary and then repeat.
Close DXF menu.

EXPORT DXF FILE

Spontaneous users

Determine what the exchange partner requires exactly: dimensioning, hatches, text, plan graphics (trees, etc.)?

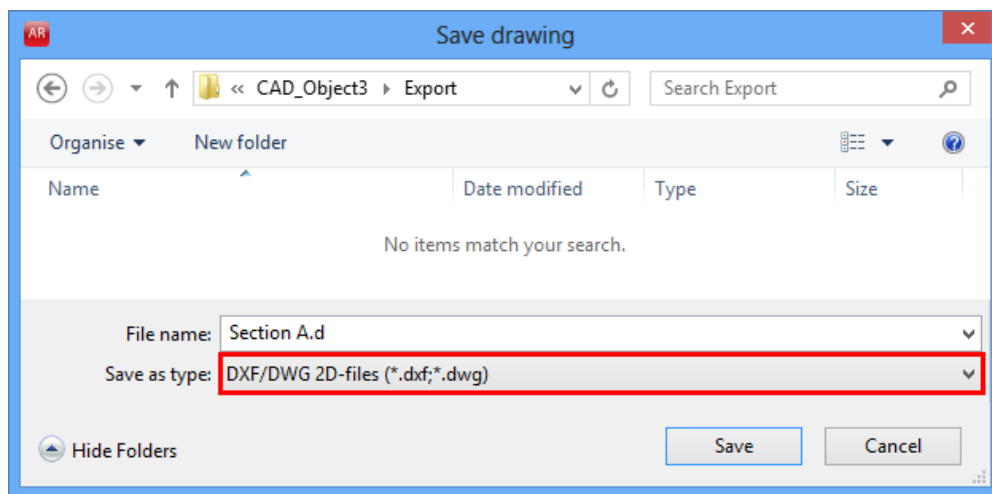
1. Prepare plan for DXF export.

2D: The export should always be made from the plan views or from a plot compilation.

3D: The export must always be made from the model area.

2. Save DXF file via the ELITECAD main menu

File > Save as



3. Set the save location, select a new file name and save.

Ensure that the file type is set correctly.

DXF/DWG 2D files (*.dxf;*.dwg) / DXF/DWG 3D files (*.dxf;*.dwg)

Comment:

The standard configuration "Dxf.dxd" is always used for this procedure.

4. Create protocol (delivery note)!

Forward DXF file (compressed if necessary).

DXF menu

Determine what the exchange partner requires exactly: dimensioning, hatches, text, plan graphics (trees, etc.)?

1. Prepare plan for DXF export.

2D: The export should always be made from the plan views or from a plot compilation.

3D: The export must always be made from the model area.

2. Open the DXF menu in the ELITECAD main menu

File > Interfaces > DXF/DWG ...

3. Import configuration

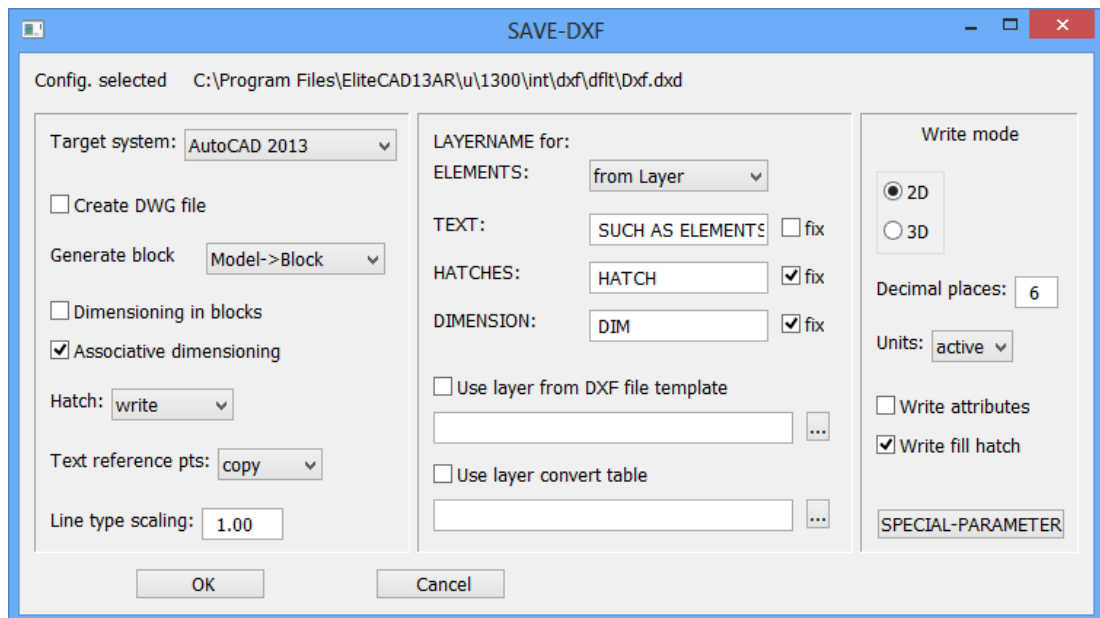
DXF - ELITE: File > Load configuration...

4. Select and open the desired configuration

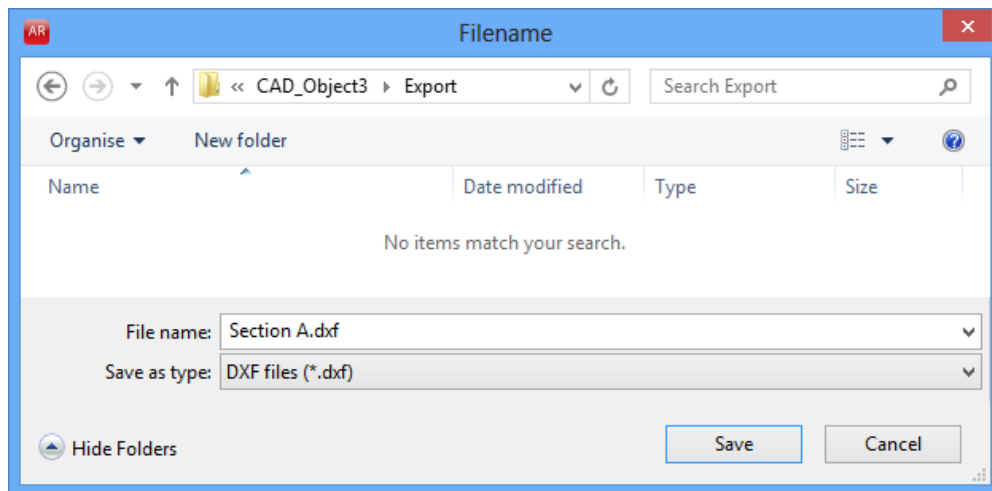
If necessary, the tables ([pages 38-41](#)) can now be adjusted using the subsequent declarations.

5. Save DXF file

DXF - ELITE: File > DXF EXPORT ...



6. Check parameter, adjust if necessary, and then click OK.



7. Set the save location, select a new file name and save.

8. Check the info window.

Interpret messages, correct procedure if necessary and then repeat.
Close DXF menu.

9. Create protocol (delivery note)!

Forward DXF file (compressed if necessary).

TROUBLESHOOTING

Problem

Although data is imported, nothing is visible.

Solution

The DXF file probably originates from a geometer. The source of the coordinates matches the system of the Swiss Federal Office of Topography.

There are various ways to resolve this problem:

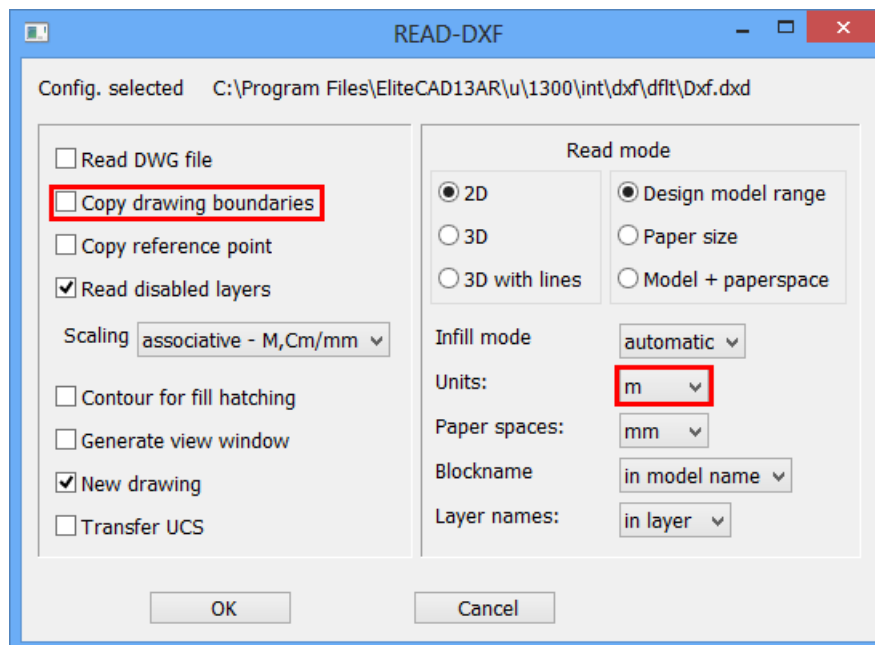
1. Option

Request a DXF with coordinate's zero-point within the drawing limits, and specification of the unit of measure.

2. Option

Try different settings in the import parameter:

Deactivate "Copy drawing boundaries" and change the units from "active" to "m".



3. Option

Search for drawing on the screen.

Simultaneously press the [Ctrl]-, [Shift]- and [Space] keys (full-screen zoom). The drawing can be very small and usually appears at the top right. Move the cut-out into the centre of your format (Ctrl+1).

Scale the plan up to its correct size, set up the relevant parts and save it as an ELITECAD drawing.

4. Option

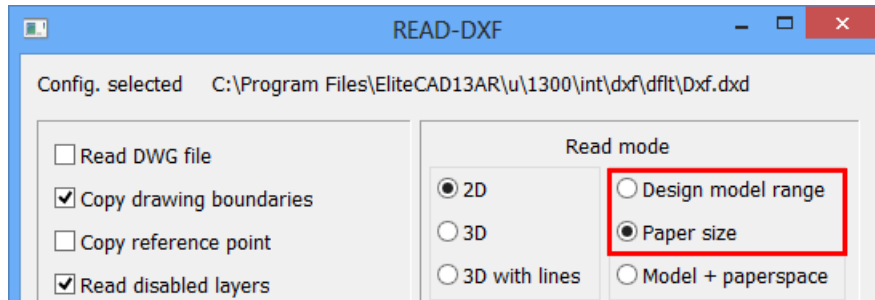
Contact our hotline.

Problem

The drawing is not complete.

Solution

Try to import the DXF file into the design model range and into the paper space.



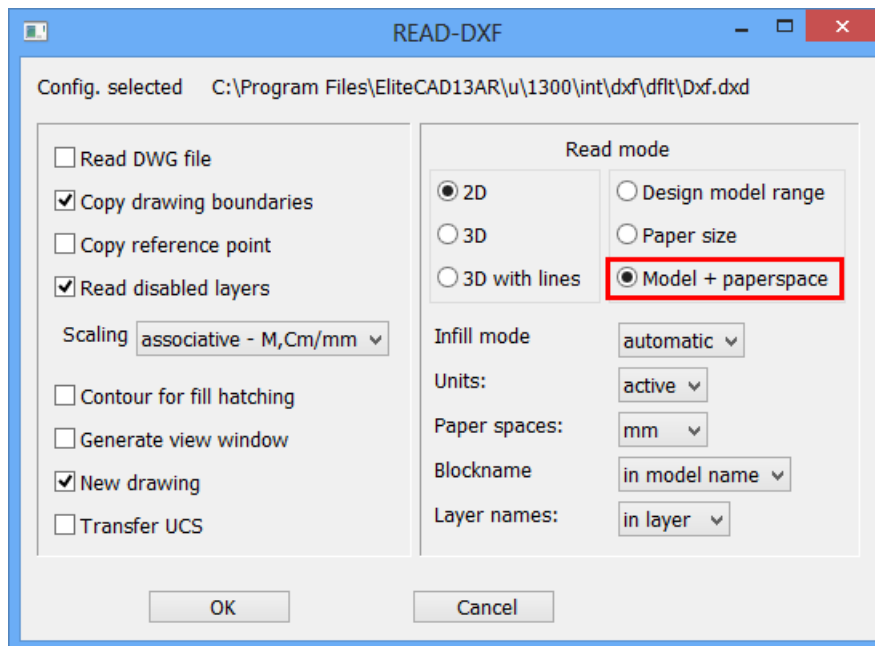
If parts of drawings exist in both ranges/spaces, the data exchange was badly prepared by your partner.

1. Option

Request a DXF in which "everything" is included in the design model range.

2. Option

You must import both ranges with model + paper range.



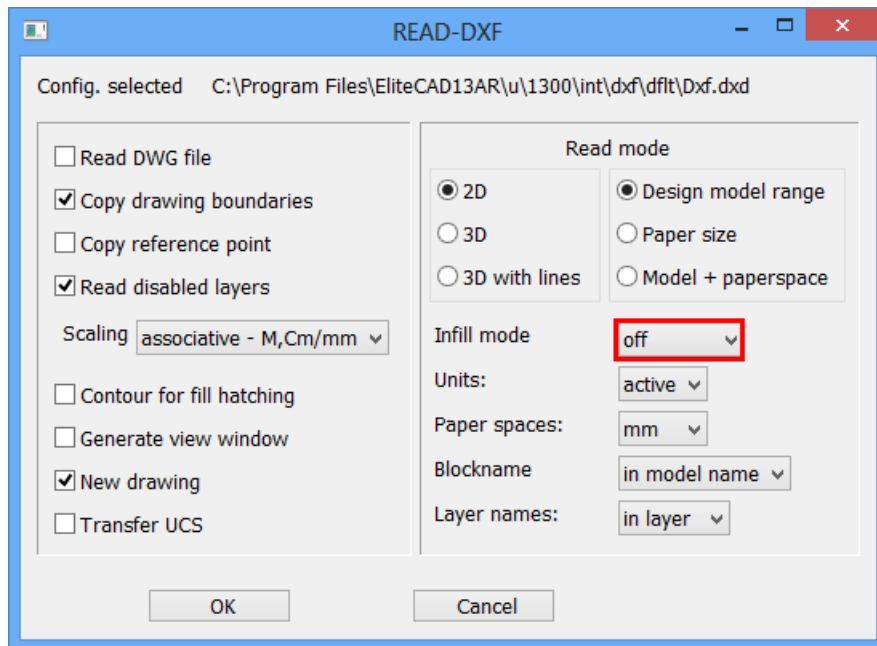
Problem

Parts of the drawing behave like hatches.

Solution

The other system uses filled out poly lines to display pen gauges. Set the fill mode to "off".

It is then possible that no more infill hatches will be imported.

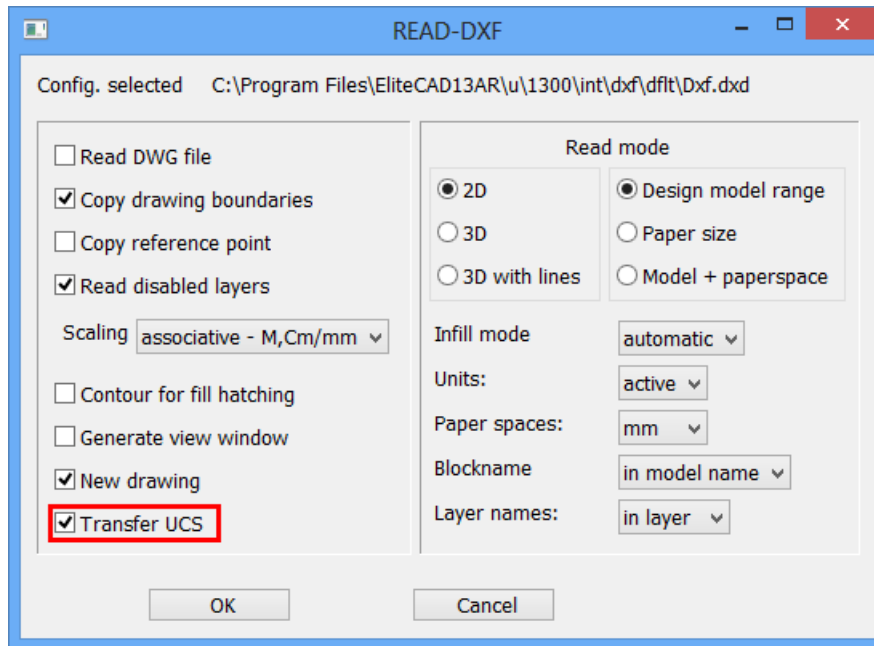


Problem

Following import, everything is displayed on one line.

Solution

The DXF was saved from a separate coordinates system (German BKS, English UCS). Activate the option "Accept UCS Coordinates System".



Other problems and solutions

Problems	Solutions
Pen colours are incorrectly displayed	Set the import table of the pens correctly. (Page 32)
Line types are wrongly displayed	Set the import table of the line types correctly. (Page 33)
Special characters are either not displayed or displayed incorrectly	Set the import table of the characters correctly. (Page 34)
Fonts are displayed wrongly	Set the import table of the fonts correctly. (Page 35)
Infill hatches and infill colours are wrongly displayed	Set the import table of the infill colours correctly. (Page 37)

Messages	Possible reasons and countermeasures
Warning: No segment found!	The file is recognised as a DXF file but the definition of the sections is either incomplete or erroneous. The drawing may be (partially) imported anyway. Request the DXF file again. If the result is the same, contact our hotline.
Line type ... not defined!	Although the line type is missing from the table it is still imported "continuously". Enter line type ... in the table. (Page 33)
Terminated ... with error!	Error codes are described in the following section.
File ... could not be opened!	Although the file is recognised as a DXF file, it is defective. No authorisation or too little storage space (DXF is buffered during import). Check these options.
End-Of-File prior to logical end - > error	DXF file is not complete or not correctly defined. If necessary, convert UNIX/DOS or MAC/DOS.

Messages	Possible reasons and countermeasures
DXF colour ... unknown, set to pen...	Pen missing from the table and will be imported as pen "DEF". Enter pen ... in the table. (Page 32)
ATTENTION: Dimension line ... was created as normal line!	Indication that the dimension line is not generated associatively. Depending on the system and setting.
ATTENTION: Dimension line ... was not created!	Problems with dimensioning definition. Correct dimension or omit if necessary.
DXF infill colour ... unknown, to infill colour ...	Infill colour missing from table and is imported as infill colour "DEF". Enter infill colour ... in the table. (Page 37)
No DXF style assigned to font ...	Font is not separately assigned in the table, becomes font "txt". Assign font ... in the table. (Page 35)
No character length table for font ... (...)	The relevant font is not available. Allocate font ... to a different (similar) ELITECAD font. (Page 35)
Characters with code ... not in font ... !	The character in question is not available in this font. These are special characters rarely found in a drawing. There are a variety of options, depending on the importance and amount of work in question: - Assign character ... to the table. (Page 34) - Change the characters in the plan manually. - Assign a different font.
Invalid contour, hatch ignored!	Message when writing infill hatches for target system AutoCAD 12 and earlier.
Hatch contours are bisecting themselves!	Message when writing infill hatches for target system AutoCAD 12 and earlier.
Text size too small! Text was not created!	Text size is too small such that the text cannot be displayed by the system. Parameter Import (Page 23): Deactivate copying of drawing boundaries And change the units from "current" to "m". Reimport.

Error codes	Possible reasons and countermeasures
9000 Error writing the file	No authorisation, insufficient storage space.
9001 Error opening file	No authorisation, insufficient storage space. (DXF temporarily stored during import).
9002 Block not found	DXF file is not correctly defined. If necessary, convert UNIX/DOS or MAC/DOS.
9003 End of file	DXF file is not complete or not correctly defined. If necessary, convert UNIX/DOS or MAC/DOS.
9005 Dimensioning point not found	Problems with dimensioning definition. Correct dimension or omit if necessary.
9006 Nesting error	Converter cannot fully interpret DXF file. Error search by specialists.
9008 Dimension invalidly defined	Problems with dimensioning definition. Correct dimension or omit if necessary.
9009 DXF – cancelled	Message following intentional or system-induced cancellation. Error search by specialists.
9010 Dimension has too many tolerances	Problems with dimensioning definition. Correct dimension or omit if necessary.

CONFIGURATION

The standard configuration is managed in the following file:

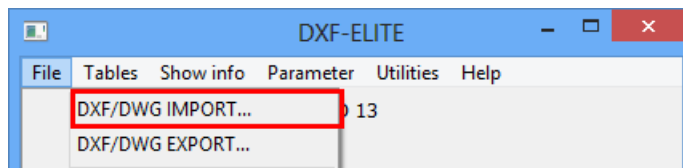
<ELITECAD-installation directory>\u\ <version>\int\dxf\dflt\Dxf.dxd

If changes are made, this file should be saved beforehand (example: Dxf_org.dxd).

Normally this standard configuration is considered to be a good setting. The majority of DXF files can then be imported without any problems. The export also functions without problems.

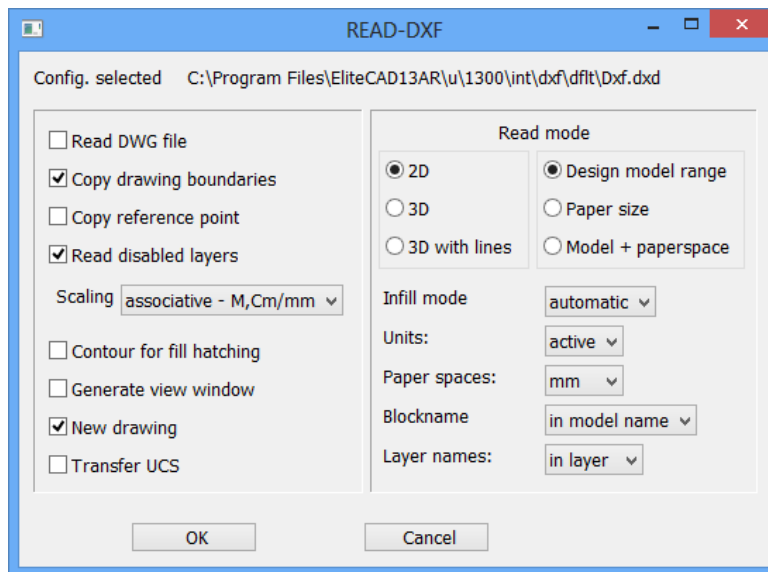
The settings apply during direct opening and saving in ELITECAD without the interface menu DXF-ELITE.

PARAMETER IMPORT



This function is used to import a DXF/DWG file taking into account the configuration settings made.

After activating the function the following settings screen appears:



Read DWG file

ON: During file selection only DWG files (*.dwg) are selected. The selected file is automatically converted into a DXF file (*.dxf).

OFF: During file selection DXF files (*.dxf) are displayed.

Copy drawing boundaries

ON: The maximum values saved in the DXF file are applied to the drawing extension. These drawing limits determine the scale that is set if the active scale is too small.

OFF: The drawing boundaries are automatically calculated.

Copy reference point

ON: The source of the drawing saved in the DXF file is used.

OFF: The drawing is centred.

Read disabled layers

ON: Layers that are marked and saved in DXF as frozen (meaning: hidden) are imported.

OFF: Only "visible" layers are imported.

Scaling

Associative - M,Cm/mm. Dimensioning including exponents are transferred fully associatively.

associative: The dimensioning is applied fully associatively. Dimensions which are not available in ELITECAD may be converted incorrectly.

free: Non-associative dimensioning is generated. This is handled entirely as dimensioning but the block stretch is not corrected.

Lines + Text: Dimensions are converted to lines and texts.

Contour for fill hatching

ON: A contour is drawn around the infill hatch.

OFF: No contour is drawn around the infill hatch.

Generate view window

ON: When the paper area is read, the view windows are generated.

OFF: No view windows are generated.

New drawing

ON: A new drawing is started prior to import.

OFF: The import of data occurs in the active drawing.

Transfer UCS

ON: The user-defined coordinates system is used.

OFF: The user-defined coordinates system is not used.

Read mode	
<input checked="" type="radio"/> 2D	<input checked="" type="radio"/> Design model range
<input type="radio"/> 3D	<input type="radio"/> Paper size
<input type="radio"/> 3D with lines	<input type="radio"/> Model + paperspace

2D: Only 2D data is imported.

3D: Both 2D and 3D data is imported.

3D with lines: 3D information which does not create 3D objects (for example, height markers) is also taken into account.

Design model range: If this check box is ticked, the design model range is imported.

Paper space: If this check box is ticked, the paper space is imported.

Model + paper spaces: If this check box is ticked, the design model range and all present paper spaces can be read. Paper spaces are automatically converted to plots and listed in views management.

Infill mode

automatic: depending on the setting for ACAD

ON: Infill hatch is automatically generated.

OFF: No infill hatch is created.

Units:

Since no units are stored in the DXF file, the correct unit for the model area can be selected here.

active: The units set in CAD are used.

mm: Data is read in millimetres.

cm: Data is read in centimetres.

m: Data is read in metres.

Paper spaces:

The correct unit for the paper area can be selected here.

active: The units set in CAD are used.

mm: Data is read in millimetres.

cm: Data is read in centimetres.

m: Data is read in metres.

Blockname

in model name: The block names are used as model names.

in attr. block: The block names are saved in the "block" attribute.

do not apply: The block names will not be used.

NOTE

If the "In attr. block" option is selected, an attribute file needs to have been loaded beforehand.

Start the "Attribute-parameter" function from the "Attributes" tool bar, enter the following path into the input line `<ELITECAD-installation directory>\u\ <version>\int\dxf\dflt\de\dxf.attr` and confirm with [Enter]

Layer names:

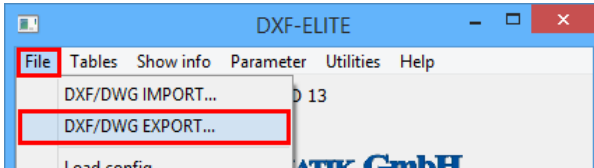
in group: The layer name is attributed to the order GROUP.

in class: The layer name is attributed to the order CLASS.

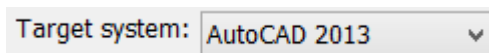
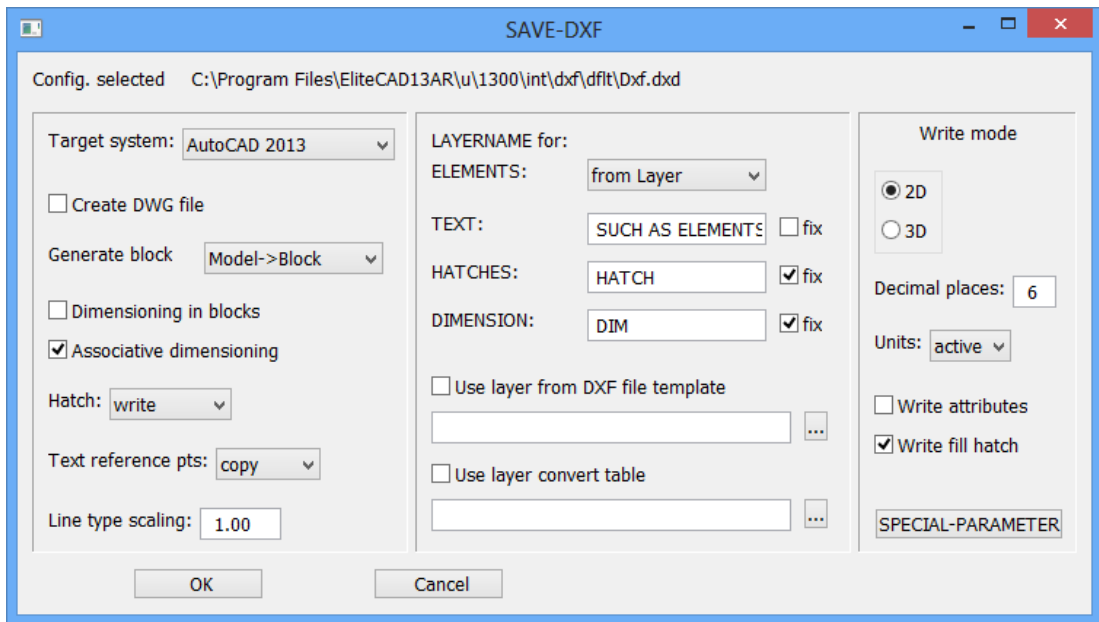
in level: The layer name is attributed to the order LEVEL.

in layer: The layer name is saved in the layer attribute and is automatically available in the Layer Manager.

PARAMETER EXPORT



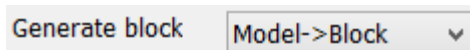
This function is used to write the current drawing as DXF/DWG file, taking into account the following settings parameters:



The desired AutoCAD/DXF version needs to be entered here. The generated data can be adjusted to the specified DX version. Clarify beforehand which version is used by your exchange partner; the transfer is optimised with the correct setting!



ON: When selecting a file, only DWG files (*.dwg) can be selected. When saved, a DXF file is generated which is automatically converted into a DWG file (*.dwg).
OFF: For the file selection only DXF files (*.dxf) are displayed and only one DXF file is generated.



Model-> Block: Every model becomes a block and the model name becomes the block name.
Level -> Block: As above except that the block name is taken from the level.
Class -> Block: As above except that the block name is taken from the class.
none: No blocks are created. (Exception: dimension blocks)

Dimensioning in blocks

ON: The dimensioning (dimension blocks) is entered into the model blocks (not very common for DXF format).

OFF: The dimensioning blocks are generated as separate blocks.

Associative dimensioning

ON: DXF dimensions (DIMENSION) are generated.

OFF: Blocks with geometry and text are generated.

Special case: If the target system is set to *easy geometry* and the check box *NO* for *Generate block* is ticked, then no blocks will be generated for the dimensions either. This setting is especially for data transfers to CAD systems that have problems with blocks.

Hatch:

don't write: Line hatches will not be written.

write: Up until AutoCAD 11/12, line hatches are written as lines (warning: files can be very large). As of AutoCAD 2000/2002, line hatches are written as associative hatches.

as block: The line hatches are written as blocks.

Text reference pts:

apply: The ELITECAD reference points (1-9) are applied in DXF.

adjusted: The text is adjusted between two points, i.e. the overall text spacing is always the same. For this reason, changes made to the text length always affects the text parameters.

aligned: The text parameters always remain the same, regardless of the text length. In other words, if a text becomes longer due to changes made, it will also require more space.

Line type scaling:

The scaling factor of the line types can be defined here.

LAYERNAME for:

ELEMENTS: ▼

TEXT: fix

HATCHES: fix

DIMENSION: fix

LAYER NAME for ELEMENTS:

from Group: The layer name is read from the order GROUP.

from Class: The layer name is read from the order CLASS.

from Level: The layer name is read from the order LEVEL.

from Layer: The layer name is read from the layer ATTRIBUTE.

LAYER NAME for TEXTS/HATCHES/DIMENSIONS

fix OFF: Layer names for texts such as elements from the layer or attribute are read.

fix ON: The content of the text fields is used for the layer name.

Use layer from DXF file template

...

Use layer convert table

...

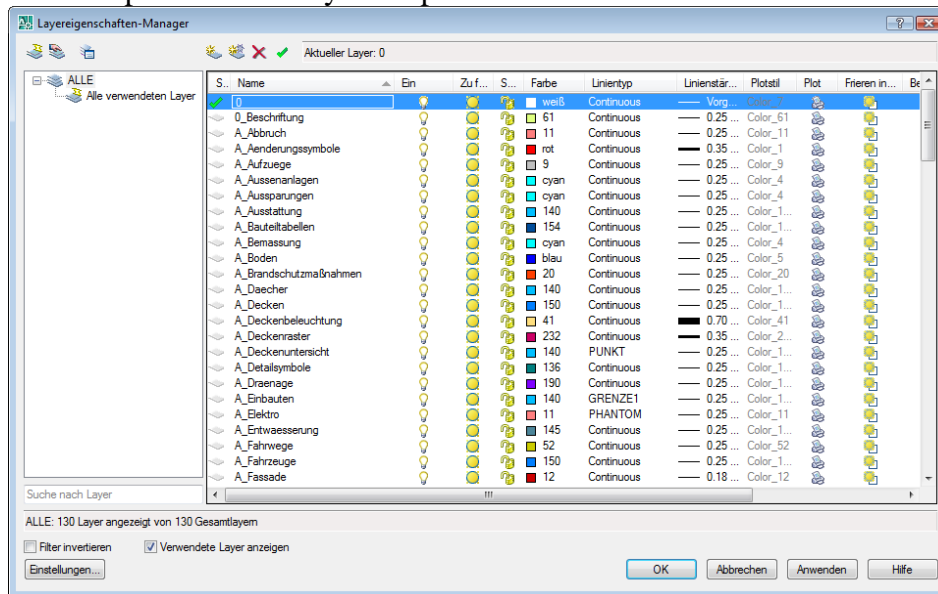
The "Use layer from DXF file template" option can be used in order to add an additional layer from the template to the DWG file.

If the "Use layer convert table" option is activated, the properties "Colour from layer", "Line type from layer" and "Line thickness from layer" are additionally added to a DWG file.

ON: A selected DXF file is used as a template for the conversion.

OFF: The properties of the individual object are transferred.

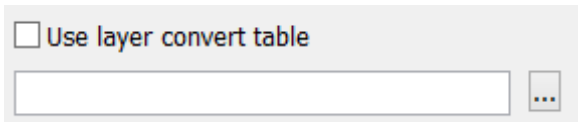
An example of a DXF layer template:



EXPLANATION

In AutoCAD, colours, line types and line thickness are often governed by the layer properties manager and not by the object itself. If these layer properties are desired by the recipient, this option can be used here.

The same layers must be defined as such in ELITECAD or be assigned as such in the conversion table. If the template does not contain a layer from ELITECAD, a notification will appear in the info window. This layer is then treated as a normal layer and is not assigned the "from layer" option.



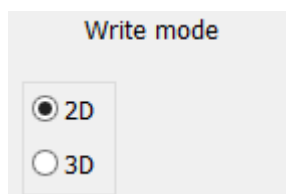
With the help of this option and an Excel file, the layers from the construction parts can be further divided, either in a different layer or with an extension to a layer name.

ON: A selected Excel file is used as a template for the conversion.

OFF: No additional layer conversion is carried out.

An example file (layerconv.xls) can be found under
 <ELITECAD-installation directory>\u <version>\int\dxf\dflt\.

	A	B	C	D
1	Command	Layer	Pen,Linetype	Destination-Layer
2	WallHatch			<>_FIL
3	WallUncut			<>_ANS
4	Hatch	212_ABBRU		<>_FIL
5	Text	212_ABBRU		<>_TXT
6	Text	003_DETAI		<>_TXT
7	Text	004_AENDR		<>_TXT
8	Hatch	005_GRAFK		<>_FIL
9	Text	005_GRAFK		<>_TXT
10	Hatch	212_ABBRU		<>_FIL
11	Text	212_ABBRU		<>_TXT
12	Text	300_BAUKO_BES		<>_TXT
13	Text	300_BAUKO_HOZ		<>_TXT
14	Text	300_BAUKO_MWK		<>_TXT
15	Element	300_BAUKO_MWK	4.7	<>_VRD
16	Hatch	300_BAUKO_STA		<>_FIL
17	Text	300_BAUKO_STA		<>_TXT
18	Hatch	300_BAUKO_STB		<>_FIL
19	Text	300_BAUKO_STB		<>_TXT
20	Hatch	300_BAUKO_STB_BES		<>_FIL
21	Hatch	300_BAUSK_GIK		<>_FIL
22	Text	300_DBELO		<>_TXT
23	Text	300_DBHLS		<>_TXT
24	DoorAxis			300_TUERE_AXS
25	DoorText			300_TUERE_TXT
26	WindowAxis			330_FASSA_AXS
27	WindowText			330_FASSA_TXT
28	Text	350_TREPP		<>_TXT
29	Element	350_TREPP	4.7	350_TREPP_UEB
30	Hatch	386_BRAND		<>_FIL
31	Hatch	387_SICZO		<>_FIL
32	Text	461_AUFZG		<>_TXT
33	Hatch	510_GELAE		<>_FIL
34	Text	520_BEFST		<>_TXT
35	ColumnHatch			<>_FIL



2D: Only 2D data can be written.

3D: All data (2D + 3D) can be written.

Decimal places:

Specifying the accuracy of the numbers (number of decimal places) for the DXF file.

Units:

active: The units set in CAD are used.
mm: The data is exported in millimetres.
m: The data is exported in metres.

Write attributes

ON: ELITECAD attributes and layers are written as DXF attributes. The ELITECAD attributes are written as being converted 1:1, unless otherwise specified in the conversion file, i.e. the attribute *id* becomes the DXF attribute *id*.

The ELITECAD layers *Sample*, *Group*, *Class*, *Level* are converted into the DXF attributes *HD_PATTERN*, *HD_GROUP*, *HD_CLASS* and *HD_LAYER*.

OFF: No ELITECAD attributes and layers are written as DXF attributes.

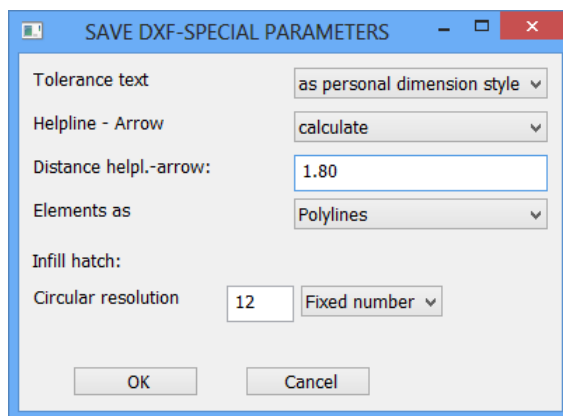
Write fill hatch

ON: Infill hatches are written as associative hatches (HATCH).

OFF: No infill hatches are written.

SPECIAL-PARAMETER

After activating the switch, the following set-up screen appears.



Tolerance text

As personal text: The two-line tolerance text is generated separately from the dimensioning block as independent text with the same layer.

As personal dimension style: For every different tolerance text, a separate DIMSTYLE (dimension parameter) is generated.

In dimension block: Two-line tolerance texts are also stored in the dimension block. The tolerance texts are lost in AutoCAD if stretched.

Helpline - Arrow

constant: The specified distance between the dimensions and the start of the dimension help line is used.

calculate: The distance is calculated from the dimension geometry and stored in the DXF dimension parameters.

Distance helpl.-arrow:

Value for the *constant* setting.

Elements as

Lines + arcs: ELITECAD elements are written as individual lines and arcs, e.g. 1 rectangle → 4 lines

Polylines: successive ELITECAD elements are written as polylines, e.g. 1 rectangle → 1 polyline

Infill hatch:
Circular resolution

(up to ACAD Release 12)

Circular resolution: Number of segments or curve length.

Fixed number: Number of segments for every full circle.

Curve length: Segment lengths in an active set unit. To avoid unwanted results, the number of segments per curve in the program is limited to a minimum of 3 and a maximum of 100.

TABLES IMPORT

Individual assignments can be set under "Tables".

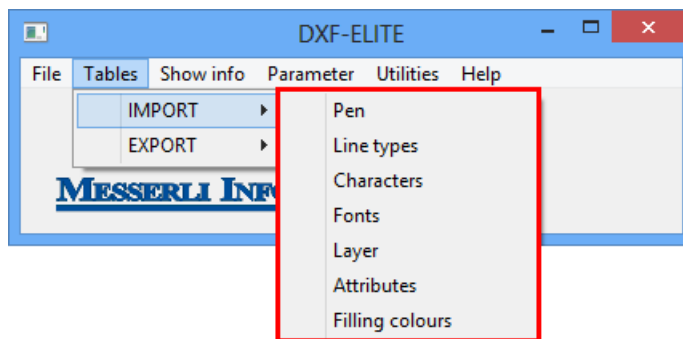
Note the following when editing tables:

- There must be at least one space between two values.
- The table is completed by an empty row.
- All entries after the empty row will be ignored.

Changes remain active only for the current session. Should you wish to retain changes long term, the configuration must be saved.

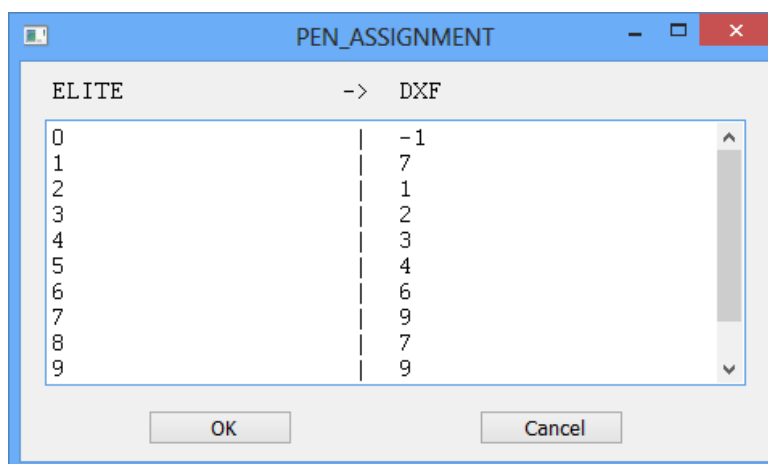
You are advised to save a configuration by name for each exchange partner.

Import pens



DXF (AutoCAD) and ELITECAD differ in the assignment of pen number and colour. In addition, ELITECAD has 1024 pens whereas DXF only has 256.

The assignment can be influenced in this table.



The DXF colour *DEF* is the ELITECAD pen for undefined DXF colours > 9.

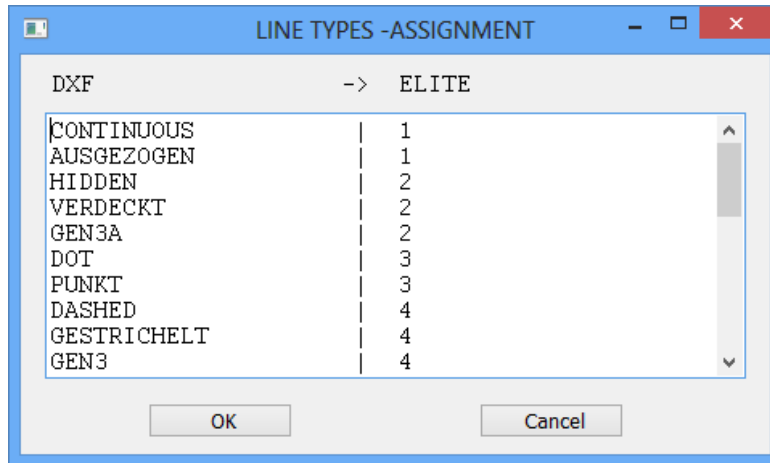
Tip:

Show info > Colours show which colours (pen numbers) are used in the DXF file.

Import line types

The standard configuration already contains the most frequent line types; in individual CAD systems however users can name line types themselves.

The assignment is determined in this table and further entries can be added to it.



DXF	ELITE
CONTINUOUS	1
AUSGEZOGEN	1
HIDDEN	2
VERDECKT	2
GEN3A	2
DOT	3
PUNKT	3
DASHED	4
GESTRICHELT	4
GEN3	4

The line type 1 is assigned to undefined line types.

Tip:

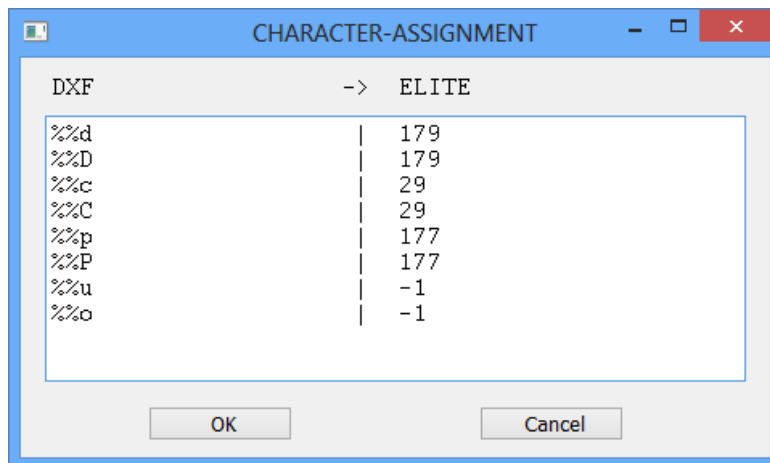
Show info > Line types show which line types are used in the DXF file (observe upper/lower case distinction).

Import characters

CAD systems work with different operating systems and character sets. "Normal" characters (code 032 – 126) are then always transferred correctly. By contrast, on older systems special characters (code 127 – 255) can be displayed in different ways. To process texts independently, special characters in DXF are defined with the prefix %%. To

In the case of newer CAD systems, only the following assignments are required.

- %%d Degree symbol °
- %%p Tolerance symbol ±
- %%c Diameter symbol Ø



The ELITECAD character "-1" (minus 1) means that the DXF character in question is deleted.

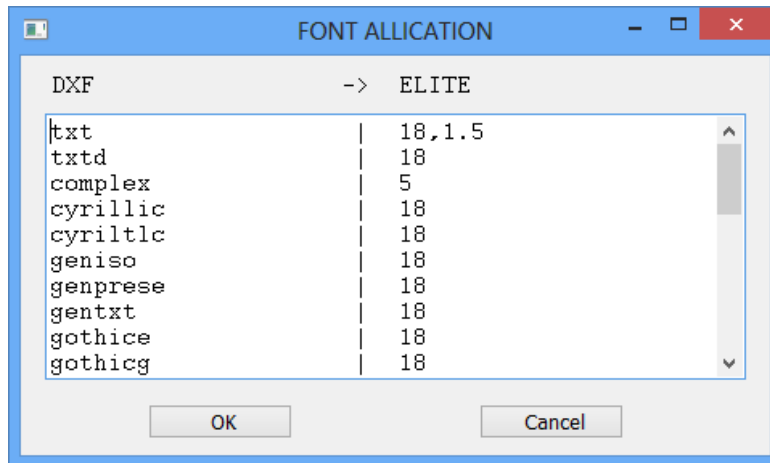
Tip:

Display info > Texts show individual texts in the DXF file.

A comparison of different character sets can be found under: Character sets ([page 44](#)).

Import Fonts

This function is used to allocate DXF fonts to ELITECAD fonts with optional character width (space), separated from the font number by a comma.

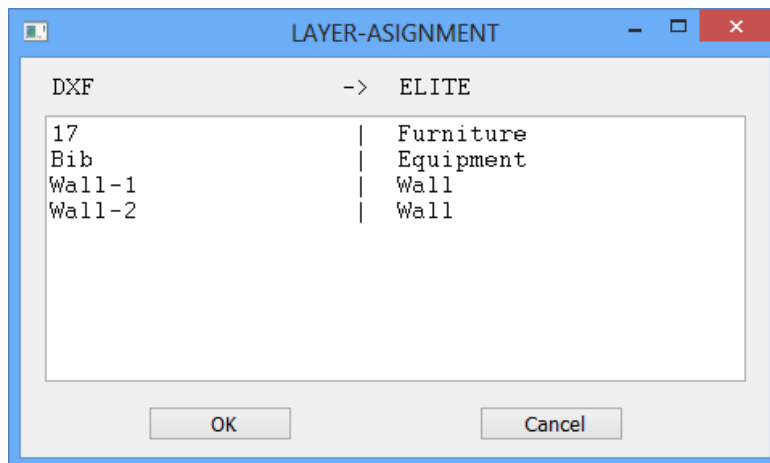


Tip:

Info > Fonts shows which fonts are used in the DXF file.

Import Layer

This function is used to assign the drawing DXF layer to ELITECAD layer.

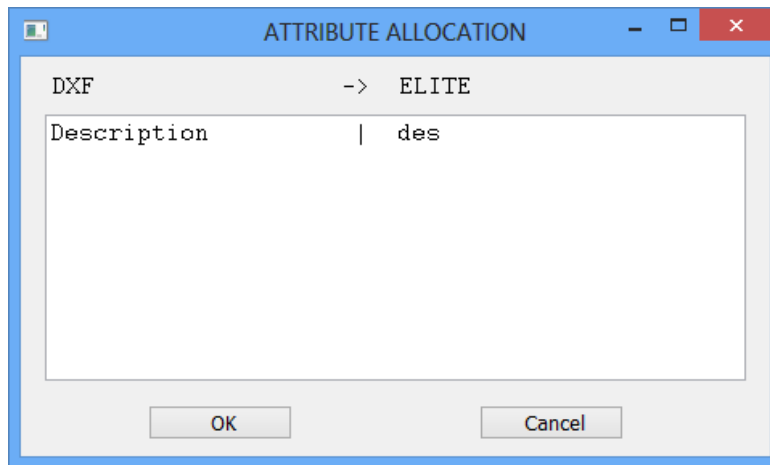


Tip:

Show info > Layers shows which layers are defined in the DXF file.

Import Attributes

This function is used to assign DXF attributes to ELITECAD attributes.



Tip:

Show info > Attributes shows which attributes were used in the DXF file.

Import filling colours

For the pens the filling colours (colour-coded hatches) for DXF (AutoCAD) and ELITECAD differ in the assignment of number and colour.

This function is used to allocate the filling colours.

TABLES EXPORT

Individual assignments can be set under "Tables".

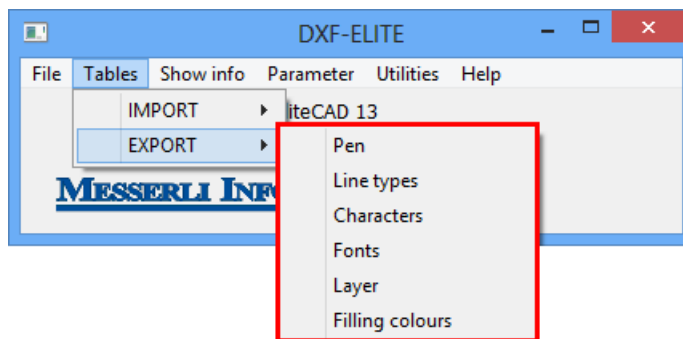
Note the following when editing tables:

- There must be at least one space between two values.
- The table is completed by an empty row.
- All entries after the empty row will be ignored.

Changes remain active only for the current session. Should you wish to retain changes long term, the configuration must be saved.

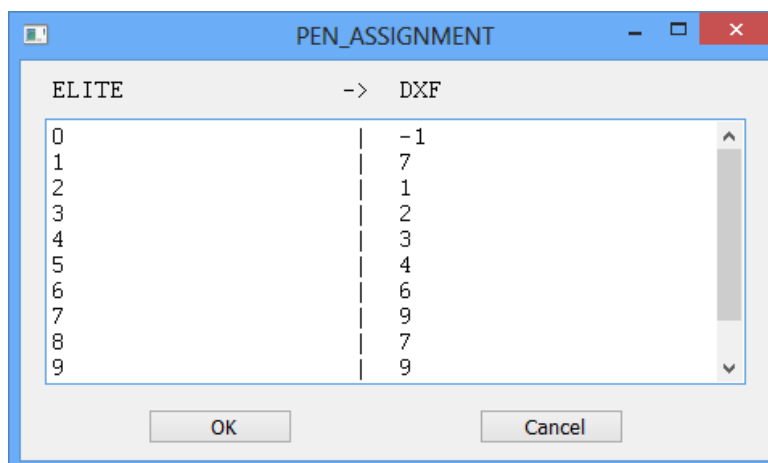
You are advised to save a configuration by name for each exchange partner.

Export pens



DXF (AutoCAD) and ELITECAD differ in the assignment of pen number and colour. In addition, ELITECAD has 1024 pens whereas DXF only has 256.

The assignment can be influenced in this table.

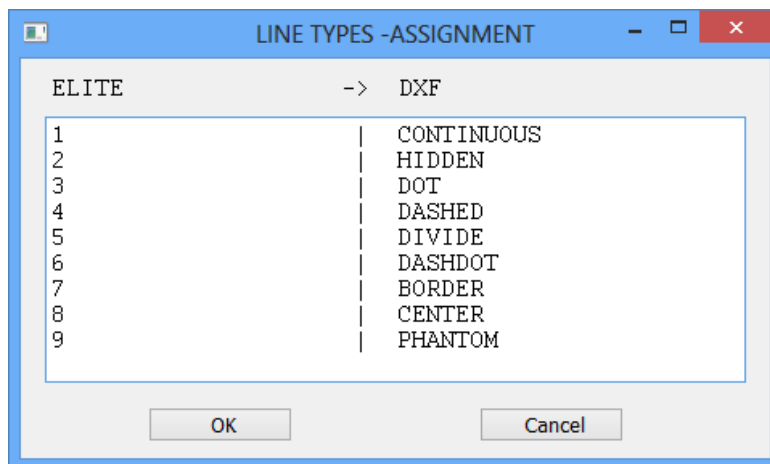


Tip:

The DXF pen "-1" (minus1) means that the ELITECAD pen in question is not written to the DXF file.

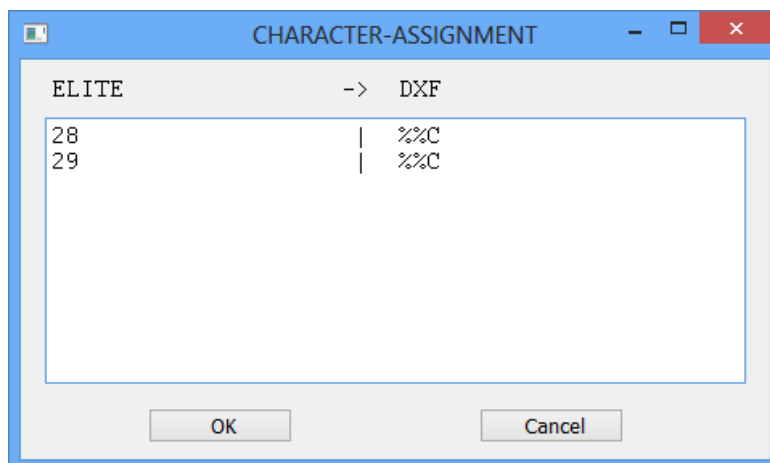
Export Line Types

This function is used to assign ELITECAD line types to the DXF line types.



Export Characters

This function is used to assign ELITECAD special characters to DXF test sequences (%%..) and to other special characters.

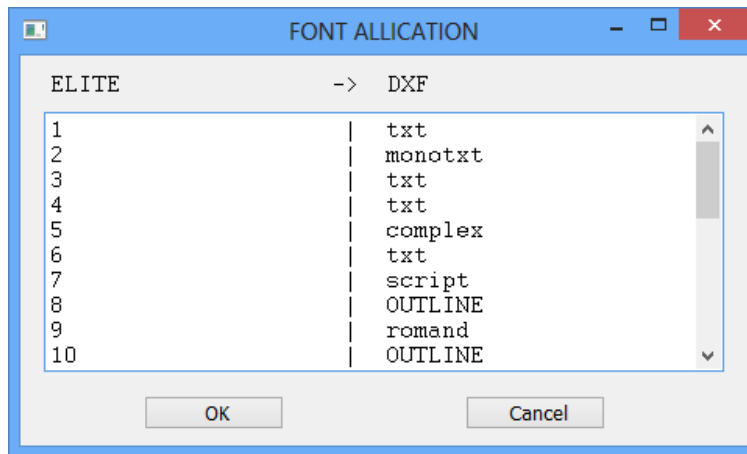


Tip:

"^5" stands for exponent 5 in the M,cm/mm dimensioning. Numbers 1 - 9 can be used here.

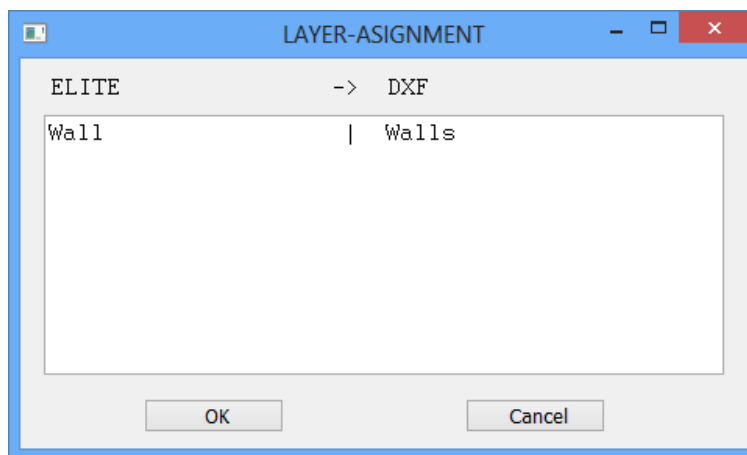
Export Fonts

This function is used to assign ELITECAD fonts to DXF fonts.



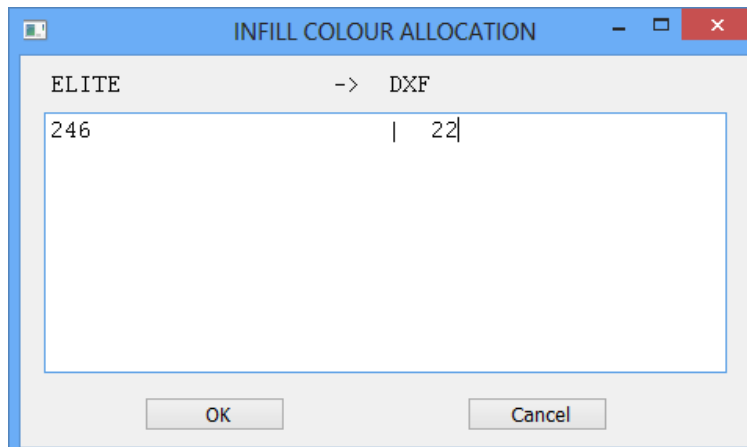
Export Layer

This function is used to assign the ELITECAD order to DXF layers. After activating the function, the following set-up screen opens.



Export Filling Colours

This function is used to allocate the filling colours.



SPECIAL CONFIGURATION

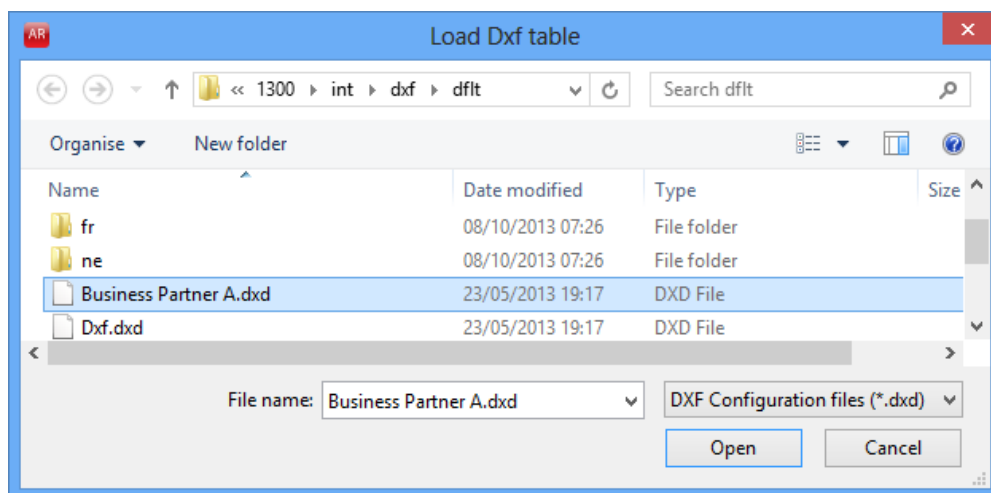
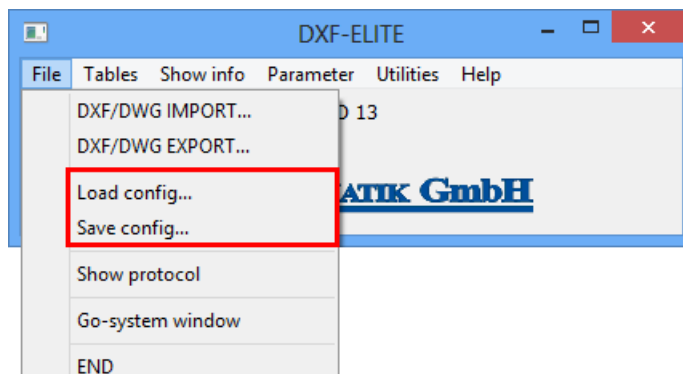
Requirement

If DXF data is exchanged with the same partner multiple times, and the settings differ from the standard configuration, it makes a great deal of sense to create a specially adapted configuration.

Create configuration

1. Open the DXF menu in the ELITECAD main menu
File > Interfaces > DXF/DWG ...

2. Import configuration
DXF - ELITE: File > Load configuration...

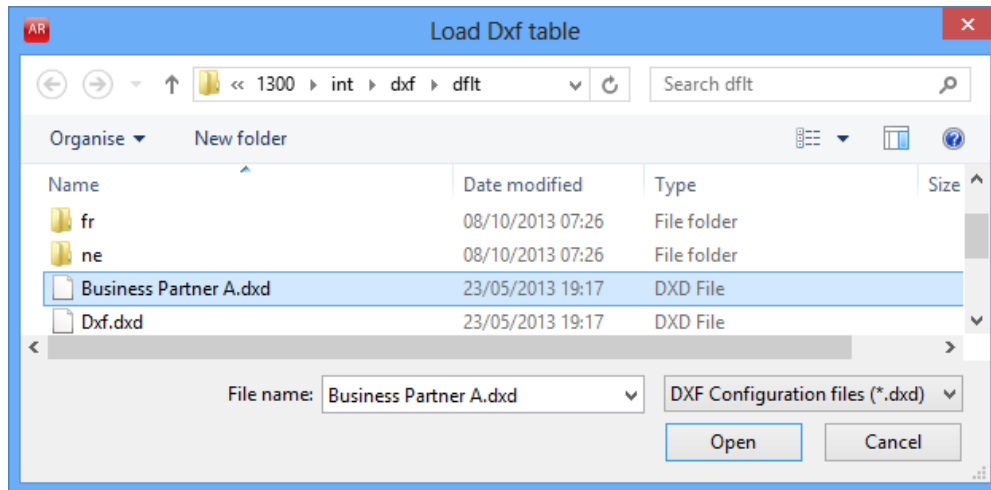


3. Select and open an existing, similar configuration.
The parameters and tables can now be adapted using protocol, data organisation, test drawing and the preceding declarations.

4. Save configuration
DXF - ELITE: File > Save configuration...

Load configuration

1. Open the DXF menu on the main menu of ELITECAD
File > Interfaces > DXF/DWG ...
2. Import configuration
DXF - ELITE: File > Load configuration...



3. Select and open the desired configuration.

Comment:

"Dxf.dxd" is the standard configuration.

A newly selected configuration always remains active until the next selection, including if ELITECAD is restarted.

CHARACTER SETS

ANSI - Table ELITECAD (Windows)

	0000	0010	0020	0030	0040	0050	0060	0070	0080	0090	0100	0110	0120	0130	0140	0150	0160	0170	0180	0190	0200	0210	0220	0230	0240	0250	
0					(2	<	F	P	Z	D	n	x	,	œ	-		*	'	¼	É	Ó	Ü	æ	ð	ú	
1)	3	=	G	Q	[E	o	y	f	—	i	«	µ	¿	Ê	Ô	Ý	ç	ñ	û		
2					*	4	>	H	R	\	F	p	z	„	Ž	-	¢	¬	¶	À	Ë	Ö	Ð	é	ò	ü	
3					!	+	?	I	S]	G	q	{	…	™	£	-	·	ˆ	Á	Ě	Ő	Ě	ó	ý		
4					"	,	.	@	J	T	^	H	r		†	§	¤	©	„	Â	Ī	Ȯ	á	é	ô	þ	
5					#	-	.	7	A	K	U	_		s	}	‡	'	>	¥	™	*	ˆ	ˆ	ˆ	ˆ	ˆ	
6					\$.	.	8	B	L	V	`	J	t	~	ˆ	'	œ	ı	°	°	°	°	°	°	°	
7					%	/	.	9	C	M	W	a	K	u	ı	‰	"	§	±	»	À	Ī	Ü	ä	ı	÷	
8					&	0	:	.	D	N	X	b	L	v	€	Š	"	ž	-	²	¼/	Æ	Đ	Ú	ä	ı	ø
9					'	1	:	.	E	O	Y	c	M	w	<	*	Y	©	³	½/	Ç	Ñ	Û	á	ı	ù	

ASCII - Table ELITECAD

	000	010	020	030	040	050	060	070	080	090	100	110	120	130	140	150	160	170	180	190	200	210	220	230	240	250	
0					(2	<	F	P	Z	D	n	x	é	ı	ú	á	ı	ı	ı	ı	ı	ı	ı	ı	ı	ı
1)	3	=	G	Q	[E	o	y	á	ı	ú	ı	ı	ı	ı	ı	ı	ı	ı	ı	ı	
2					*	4	>	H	R	\	F	p	z	á	Á	y	ó	¼	Á								
3					!	+	?	I	S]	G	q	{	á	Á	Ó	ú	ı	ı	ı	ı	ı	ı	ı	ı	ı	
4					"	,	.	@	J	T	^	H	r		á	É	Ü	ñ	«	©							
5					#	-	.	7	A	K	U	_		s	ç	æ	ø	Ñ	»								
6					\$.	.	8	B	L	V	`	J	t	é	Æ	£	°									
7					%	/	.	9	C	M	W	a	K	u	ı	é	ó	ø	°								
8					&	0	:	.	D	N	X	b	L	v	Ç	é	ó	x	ı								
9					'	1	:	.	E	O	Y	c	M	w	ü	ı	ó	f	©								

ASCII - Table Windows

	000	010	020	030	040	050	060	070	080	090	100	110	120	130	140	150	160	170	180	190	200	210	220	230	240	250
0					(2	<	F	P	Z	D	n	x	é	ı	ú	á	ı	ı	ı	ı	ı	ı	ı	ı	ı
1	⊙	⊙	⊙	⊙)	3	=	G	Q	[E	o	y	á	ı	ú	ı	ı	ı	ı	ı	ı	ı	ı	ı	ı
2	⊙	⊙	⊙	⊙	*	4	>	H	R	\	F	p	z	á	Á	y	ó	¼	Á	ı	ı	ı	ı	ı	ı	ı
3	⊙	⊙	⊙	⊙	!	+	?	I	S]	G	q	{	á	Á	Ó	ú	ı	ı	ı	ı	ı	ı	ı	ı	ı
4	⊙	⊙	⊙	⊙	"	,	.	@	J	T	^	H	r		á	É	Ü	ñ	«	©						
5	⊙	⊙	⊙	⊙	#	-	.	7	A	K	U	_		s	ç	æ	ø	Ñ	»	ı	ı	ı	ı	ı	ı	ı
6	⊙	⊙	⊙	⊙	\$.	.	8	B	L	V	`	J	t	é	Æ	£	°	ı	ı	ı	ı	ı	ı	ı	ı
7	⊙	⊙	⊙	⊙	%	/	.	9	C	M	W	a	K	u	ı	é	ó	ø	°							
8	⊙	⊙	⊙	⊙	&	0	:	.	D	N	X	b	L	v	Ç	é	ó	x	ı	ı	ı	ı	ı	ı	ı	ı
9	⊙	⊙	⊙	⊙	'	1	:	.	E	O	Y	c	M	w	ü	ı	ó	f	©	ı	ı	ı	ı	ı	ı	ı