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## TechNotes and FAQs

### DWG FILE FAQ

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**Subject:** DWG File  
**Product:** MicroStation V8  
**Operating System:** Windows® 2000, Windows XP Professional, Windows XP Home Edition, Windows Me, Windows NT® 4 (SP6 recommended), Windows 98 (Second Edition recommended)  
**Document Number:** 6216

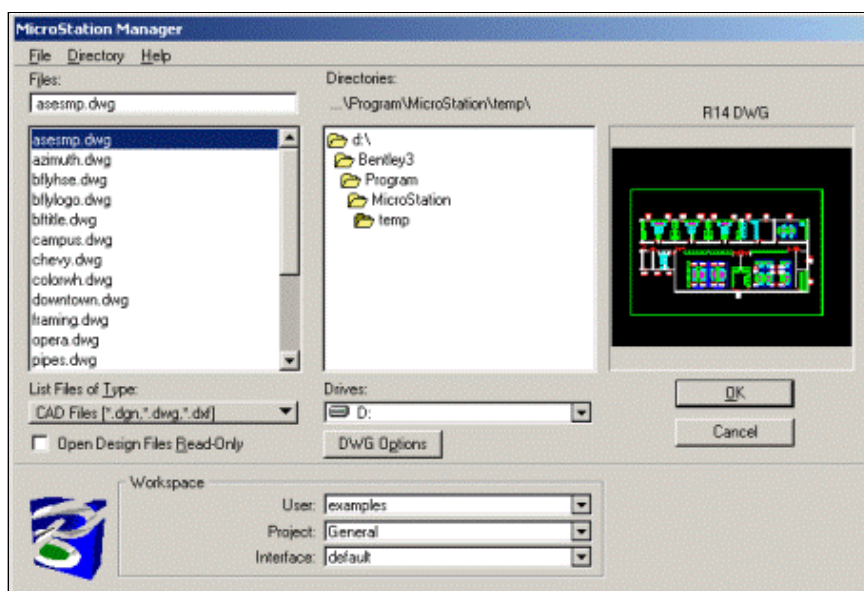
*Note:* This FAQ applies to MicroStation V8 v08.00.00.21.

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#### **Question 1: How do I open a DWG file in MicroStation V8?**

*Answer:* To open a DWG in MicroStation V8 from the MicroStation Manager, change the list files of type to CAD Files (or DWG). Highlight the file you wish to open and select OK.



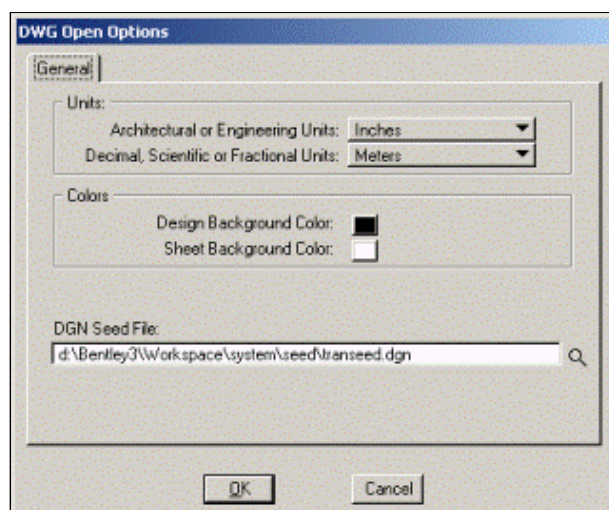
**Question 2: Can I change any settings when opening a DWG?**

*Answer:* The following options can be set specifically when working with DWGs:

1. Units - Architectural/Engineering and Decimal/Scientific/Fractional
2. View window background colors for the design model ("model space") and the sheet model ("paper space").
3. DGN seed file

To set these options, click the DWG Options button in the Open File or MicroStation Manager dialog box. This button is present only while a DWG or DXF file is selected in the Files list box.

- **Units - Architectural/Engineering and Decimal/Scientific/Fractional:** MicroStation V8 uses units so when a DWG file is opened in MicroStation V8, it must also have units. The units option will allow the user to tell MicroStation V8 what units to be used when opening a DWG file. All AutoCAD file units are listed in the dialog box to let user choose. For example, if AutoCAD file units are set to Architectural, a user should select Inches so that the result will be compatible.
- **View window background colors for the design model ("model space") and the sheet model ("paper space"):** This is the only place that the background color can be set.
- **DGN seed file:** The seed file will contain items such as the working unit labels and the design file settings.



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**Question 3: Which workmodes does MicroStation V8 support?**

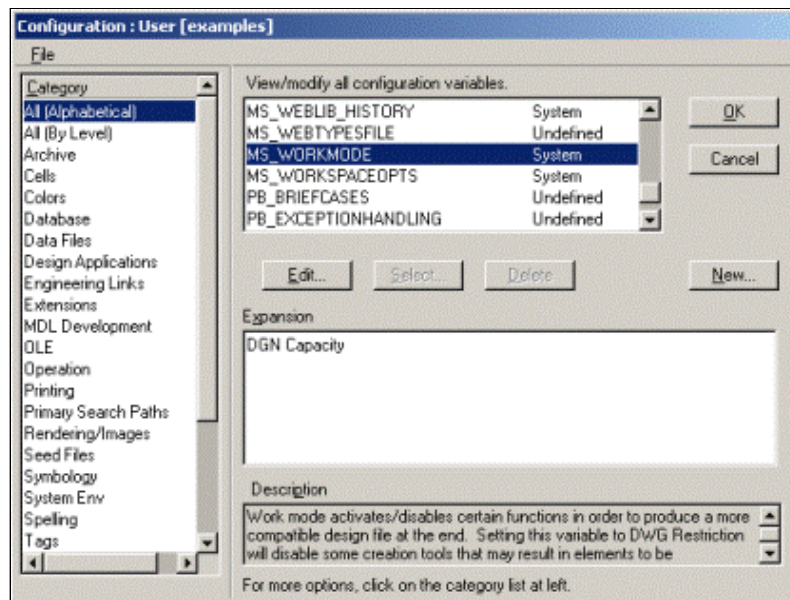
**Answer: MicroStation V8 supports the following workmodes:**

- DGN (Defines the DGN Capacity workmode)
- DWG (Defines the DWG Restriction workmode)
- V7 (Defines the V7 workmode)

The current workmode is specified by the configuration variable MS\_WORKMODE. However, when you open a DWG file, DWG mode is automatically enabled regardless of the MS\_WORKMODE value. While DWG mode is enabled, an icon is displayed in the status bar. The icon depicts a white crosshair on a red background.

**To set the workmode:**

1. From the Workspace menu, choose Configuration. The Configuration dialog box opens.
2. In the alphabetical list of configuration variables, select MS\_WORKMODE.
3. Click Edit. The Edit Configuration File dialog box opens.
4. In the New Value field, key in DWG Restriction or DGN Capacity.
5. Click OK. The focus returns to the Configuration dialog box.
6. From the dialog box's File menu, choose Save.
7. Click OK.
8. Exit and restart MicroStation V8.



**Question 4: What are the restrictions to the DWG workmode?**

*Answer:* Due to limitations in the DWG format, the DWG workmode ensures maximum DWG format capability by restricting certain MicroStation V8 functionality from creating information that can not be stored in DWG. Note that simply changing workmodes will not modify existing geometry. Only future file modifications are affected.

The following list summarizes the MicroStation V8 limitations while in DWG workmode:

- **Colors:** The DWG color palette is the only color table loaded; no other color tables can be loaded. The menu item Settings > Color Table is disabled. An exception to this is that you can still change the color palette by importing raster files. If you choose to use the color palette of the raster file - that is, you turn off Current Color Palette - the color palette changes.
- **Line styles:** Line styles 1-7 are disabled and only custom line styles are allowed.
- **Element class:** The Active Class is set to Primary and cannot be changed.
- **View window background color:** The Black Background > White preference in the View Options category of the Preferences dialog box (Workspace > Preferences...) is disabled. To set the display for models and layouts of DWG files, use the DWG Options dialog box just before opening a DWG or DXF file.
- **Grid orientation:** The grid is aligned with a Top view such that it extends along the X and Y axes from the global origin. To rotate the grid about its Z-axis, key in ACTIVE GRIDANGLE <rotation\_angle> or define and rotate an ACS.
- **References:** The active model cannot be self-referenced. A reference cannot be clipped by mask.
- **Dimensioning:** The following dimension settings and capabilities are not supported:
  - Custom prefixes and suffixes
  - Dimension line level
  - Dimensioning at an "arbitrary" position
  - Terminator-specific symbology
  - Terminator character symbol
  - First and joint terminator types for consecutive dimensions
  - Custom units
  - Unit separation at thousandth and millionth
  - Secondary dimension-specific leading/trailing zeros

- Alternate labeling
- Arc symbol above text displayed at an angle
- Arc-length format for angular dimension
- Cell terminator-specific width and height

*Additional note:* All dimensions must be placed using a dimension style. The seed DWG file, "seed.dwg", contains a "Standard" style. When you open a DGN file that does not contain or refer to any dimension styles, MicroStation V8 copies the Standard style is copied to the DGN file.

**Annotating with flags:** Tools in the Annotate tool box, which are used to place and edit flags, are disabled.

**Importing raster images:** Intergraph raster images, Sun raster, procedural application files, Geo-referenced TIFF files, and Img files cannot be imported.

**Managing raster files:** Raster Manager does not support the following capabilities:

- Assigning logical names and descriptions to attached images.
- Reference agent.
- Attaching of Viecon Publisher images.
- Display of attached images on a per view basis.
- Scaling images non-proportionally.
- Mask clipping.
- Image tint control.
- Image display gamma control (this setting is fixed at 1.0).
- Image plot gamma control (this setting is fixed at 1.0).
- Image-specific printing.
- Attaching images in formats other than BMP, CALS-I, JFIF/JPEG, PCX, PNG, RLC, TARGA, and TIFF.
- Image transparency control (disabled for binary, RGB, and COT images only - settings are fixed).
- Image background color control (disabled for binary images only).
- Image inversion control (disabled for binary images only - Invert is on).

**Tags:** Tag elements can be attached only to cells.

**Complex elements:** Spline curves cannot be selected as part of a complex chain or complex shape.

**3D modeling:** 3D elements can be placed only as ACIS bodies. The Parasolid modeling engine is disabled.

**Cells:** Normal (unshared) cells are not supported in DWG. When you place a normal cell, a Type 2 cell name element is not stored in the open file. Rather, the component elements are stored as a group (also known as an orphan cell). Groups cannot be named.

**Area patterning:** Area patterning is replaced by AutoCAD pre-defined hatching. The Pattern field in the Pattern Area settings window is a combo box which, by default, lists the patterns in the DWG pattern file "areapat.pat". This file contains a DWG pattern definition for each pattern cell in the supplied cell library, "areapat.cel". To select a different DWG pattern file, key in its full file specification - path and filename - in Pattern File or click the adjacent magnifying glass icon to browse the file system.

**Dimension-driven design:** Dimension-driven design tools are disabled.

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#### **Question 5: How do I use .shx and .ttx fonts in MicroStation V8?**

*Answer:* MicroStation supports text styles and provides an interface for constructing text with available system fonts and a wide variety of text attributes. The following fonts are supported natively in MicroStation:

- MicroStation
- True Types
- AutoCAD fonts (.shx)

If AutoCAD is installed, MicroStation by default will read the AutoCAD fonts (.shx). If AutoCAD is not installed but the AutoCAD fonts (.shx) are available a configuration variable can be set. Workspace > Configuration > MS\_DWGFONTPATH. Set this to the folder containing the AutoCAD fonts (.shx).

#### **Question 6: How do I save a file as a DWG?**

*Answer:* From the Save As interface choose the "AutoCAD Drawing file" option under the List of file type option. Now select the DWG Option button.

- DWG version: This will allow you to select the version of AutoCAD you wish to save to.
- Units: The next options is units, this allows the setting of the units for the DWG file.
- Level Display: This option controls the view you would like to use for your level display when exporting the file. In other words if you have a certain level display in Window 1 and you would like to use this, you would select the Window 1 option here.
- Save Reference as a DWG/DXF: This option lets you save your reference file out in DWG or DXF format. This is beneficial especially if the file must be opened in AutoCAD.
- Another SaveAs option that is beneficial for opening the file is converting your font over to AutoCAD fonts. You can browse and find the location of the SHX files that you want to map your fonts to.

The last option allows you to select a file to specify a prototype drawing file or seed file to determine settings to the translated drawing file.

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#### **Question 7: In the Save As dialog box what is the advanced tab used for?**

*Answer:* The Advanced Tab currently includes a single setting:

- Create Regions From Closed Elements: (Never, 3D Only, Always).

This setting controls the type of AutoCAD entity that will be created from a Complex Shape, Circle, Ellipse or Shape element with more than 4 vertices. If this setting is set to "Never" (or "3D Only" and model is 2D) these entities are stored in AutoCAD files as either closed PolyLines, Circles or Ellipse entities. This is the most compact representation and is sufficient for most purposes. However, AutoCAD does not treat these entities as opaque entities for rendering purposes. In order for AutoCAD to render these as opaque, it is necessary to save them as the more heavyweight "Region" entity. A file with Region entities is typically substantially larger and slower to work with in AutoCAD, but if rendering is important it is probably desirable to set this setting to "3D Only". As 2D files are not useful for rendering, the "Always" setting is typically not appropriate.

#### **Question 8: Why does "Zoom Extents" in AutoCAD look different from a "Fit View" in MicroStation?**

*Answer:* Check the geometry on layers that are turned off. The AutoCAD "Zoom Extents" area will include layers that have display turned off. The MicroStation "Fit" tool will fit only visible geometry.

If you wish to have the AutoCAD "Zoom Extents" tool ignore geometry on a certain layer then it will be necessary to "Freeze" the layer. The AutoCAD Zoom-Extents command does not fit frozen layers.

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#### **Question 9: When saving a DGN as a DWG file, the seed file specified in the DWG Options, has dimension styles, text styles, and paper space layouts that are missing from the resulting DWG file. What is used as the source file for the new DWG file?**

*Answer:* When working in AutoCAD, template files are used to create new files; when working in MicroStation, seed files are used to create new files. In MicroStation you can use a DWG seed to create a DWG file or a DGN seed to create a DGN file. In both software packages, the resulting new file will be based on the seed/template.

This applies only to the creation of a new file using the File > New command. If you select File > Save As you still create a new file, but it isn't based on a seed/template, it is based on the file being saved.

When saving (converting) files from one format to another, you also use the File > Save As command. The content of the active file is saved to a new file that is exactly the same as what is being saved. Nothing more, nothing less.

In conversion, there can be settings that do not exist in the original format but are necessary for the resultant format. An example of this is seen when saving a DWG file as a DGN file. MicroStation has Working Units, and AutoCAD has a single drawing unit. Although similar, they are not the same. Therefore, when saving a DWG file to DGN, there are settings that will pull the Working Units for the converted DGN file from a specified DGN seed file, because a DGN file must have Working Units.

For settings such as dimension styles or text styles both files have equivalent settings, so only the dimension or text styles contained in the DWG file will be saved into the new DGN file.

There is no comprehensive list of what settings are controlled by the seed file when going from one format to another. The rule of thumb is that anything that does not exist in the original format but is required in the resultant format will be taken from the seed file. During conversion, no extra information will be created.

This usage of the seed file is a bit confusing, given that in past versions of MicroStation we had to export the information out of MicroStation into AutoCAD. When exporting we did use the seed file to create the new DWG. In MicroStation V8.5 this is not the case. The DWG file format is a subset of the DGN file format, so we are transforming existing data and do not actually create a new file.

If you have a set of standards that the resulting DWG files have to meet, such as specific dimension styles or text styles that need to exist in all DWG files, these styles can be created in a dgnlib file and used throughout the project files.

If you do not wish to use dgnlib files, the dimension and text styles can be imported into the DGN file prior to saving to DWG. Another option is to import the dimension or text styles into the DWG in AutoCAD after using the File > Save As command.

#### Question 10: Does MicroStation 2004 Edition support AutoCAD 2005 files?

*Answer:* Yes. AutoCAD 2005 and AutoCAD 2004 share the same file format, so MicroStation 2004 Edition will open AutoCAD 2005 files without any problems. MicroStation displays, but does not allow manipulation of, new AutoCAD 2005 features such as sheet sets, tables, and fields. Support for these new features is planned for the next version of MicroStation.

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