



Date July 16, 2000 Memo Number STI56:000716
Subject ANSYS Tips & Tricks: Opening ANSYS Files by Double-Clicking
Keywords Windows NT: File Association

1. Introduction:

Double-clicking ANSYS *.db files in Explorer under Windows is a very useful feature. This memo describes how to configure ANSYS 5.4 through 5.6 to enable opening database files by simply double-clicking on the *.db icon. [It is useful to note that while ANSYS 5.6 automatically enables file association with the *.db file, there are many limitations to this approach.]

2. Background Discussion:

Starting from ANSYS 5.6, the installation setup process under Windows associates *.db and *.grph files, such that they can be opened in Explorer simply by double-clicking on the *.db or *.grph icon. The way this works is that an undocumented "-FILE" argument to the "ansysir.exe" and "displayw.exe" executables is used for the file association.

While this may seem to be an improvement from previous versions of ANSYS running on Windows, the author finds this technique limiting for the following reasons:

- This ability is only available for 5.6. 5.5 has a similar undocumented option, but it does not automatically resume the database.
- The last settings specified in "ANSYS Interactive" (ansysi.exe) are used. This means that the working directory and jobname are left unchanged from the previous analysis. Subsequent SAVE commands will produce undesired results if the user is opening a different database from the last-specified jobname.
- Similar to the point above, since the last settings are used, changing memory settings (total workspace and database space) and graphics drivers (win32, win32c, 3D) are not permitted without re-launching "ANSYS Interactive" and respecifying the appropriate values.

These points make this approach cumbersome and confusing. Hence, the author finds that manually configuring Windows file association gives the user more flexibility.

3. Discussion on Association of *.DB Database Files:

The following steps can be performed under Windows to associate *.db files with ANSYS:

1. Create a DOS batch file (regular text file) named "launch_ansys56.bat" as follows:

```
-----launch_ansys56.bat-----
@echo off
set PATH=%ANSYS56_DIR%\bin;%ANSYS_SYSDIR%;%PATH%
start /b ansys -j %~n1 -p ane3flds -m %2 -db %3 -d %4 -g -SIRESUME yes
-----end of file-----
```

Save this batch file in any location, such as C:\location_of_scripts.

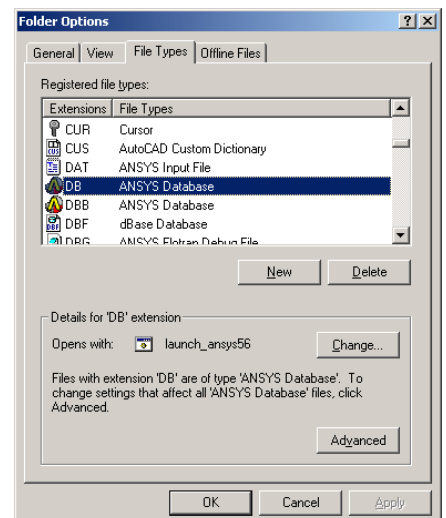
2. Modify your "start56.ans" file located in your ANSYS installation directory which, by default, is C:\ansys56\docu:

```
-----add to start56.ans-----
*get, S1STATUS, parm, SIRESUME, type
*if, S1STATUS, eq, 3, then
    *if, SIRESUME, eq, 'yes', then
        resume
    *endif
*endif
-----end of file-----
```

3. Go in Windows Explorer, under "View > Folder Options" (or "Tools > Folder Options" in Windows 2000). Select the tab "File Types". Scroll down to find the "DB" file extension (or, if listed by name, it may read "ANSYS, Inc."). Select "Edit" (or "Advanced" under Windows 2000). Edit the "Open" action, and type the following instead:

```
C:\location_of_scripts\launch_ansys56.bat "%1" 128 32 3D
```

The file extension "*.db" is now redefined to be launched by the script "launch_ansys56.bat". This, in turn, will launch ANSYS in the working directory with the correct jobname. In 3D graphics mode, the model





will be resumed and plotted (for win32/32c, the model will be resumed but not plotted).

In Step 1 above, a DOS batch file was used to parse arguments to ANSYS and launch the ANSYS executable. The second line adds the ANSYS executables, namely the “ansys.exe” program, in the PATH variable. The third line launches the ansys.exe program standalone (“start /b ansys”). The jobname is input via the %~n1 argument; usually, Windows passes the full file name to the program. The use of the %~n1 argument restricts this input to just the file, without file extension (i.e., the ANSYS jobname). The license product is specified with “-p”; in this case, “ane3flds” uses ANSYS/Multiphysics/LS-DYNA. Some typical product license codes include:

- ansys ANSYS/Mechanical
- ane3 ANSYS/Mechanical/Emag
- ane3fl ANSYS/Multiphysics
- ane3flds ANSYS/Multiphysics/LS-DYNA

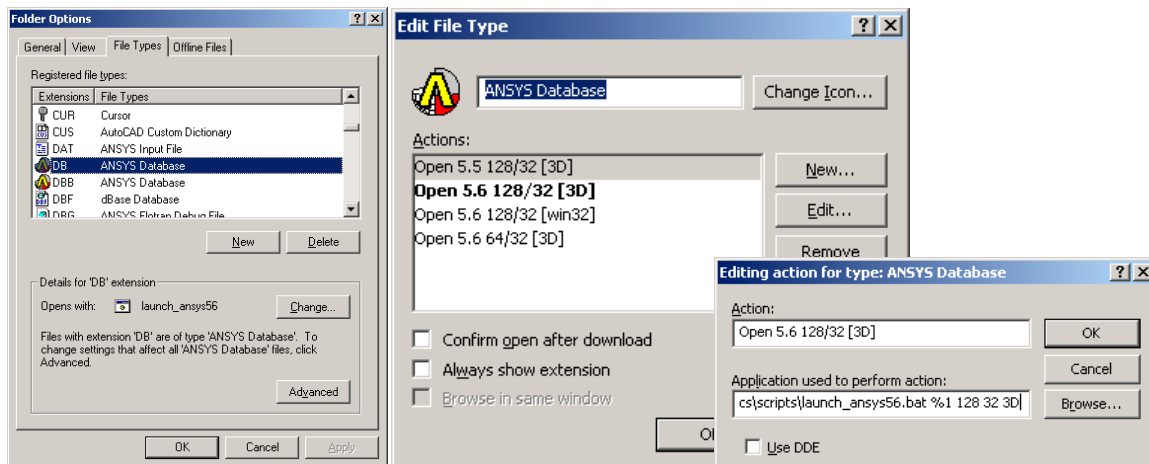
Please refer to the “ANSYS Installation and Configuration Guide” for the appropriate product code for your license. The workspace (-m), database space (-db), and graphics driver (-d) are input as arguments to the batch file (explained later). ANSYS is run in interactive mode (-g). Lastly, a parameter called “SIRESUME” is also passed to ANSYS with a value of “yes”.

In Step 2 above, the “start56.ans” ANSYS startup file was modified. Whenever ANSYS is launched, the start56.ans file is read from the C:\ANSYS56\DOCU directory.¹ This can be “forced” in both GUI and batch mode with the “-s read” argument. The startup file contains information on settings and commands which ANSYS uses each time it is launched. By adding the lines in Step 2, a check of a parameter called “SIRESUME” is performed. If “SIRESUME” is “yes”, ANSYS resumes “jobname.db”; otherwise, nothing happens.

The action “Open” (which is usually the default action) is associated with double-clicking the *.db file. In Step 3, this action was redefined to be associated with the DOS batch script created in Step 1. Furthermore, four arguments were passed to it: the jobname (“%1”), 128 MB for total workspace, 32 MB for database space, and 3D graphics driver. If different settings are desired for the “default” action, one can change this easily (e.g., change the arguments to “64 32 win32” for 64 MB total workspace, 32 MB database space, and win32 graphics driver).

4. Advanced Options for DB File Association:

To allow the user more flexibility in selecting graphics drivers, product licenses, and workspace & database memory settings, multiple actions can be specified:



Above are “Tools > Folder Options” dialog boxes for Windows 2000, although Windows NT is very similar in nature.

An icon can be selected for the *.db files. Select “Change Icon” (window in middle) and browse to search for the ansys.exe executable (by default, located in C:\ansys56\bin\intel\ansys.exe). Any other icon can be selected instead of the default generic icon.

¹ If a “start56.ans” file resides in the %HOMEDRIVE%\%HOMEPATH% directory or the working directory, ANSYS will read that file instead of C:\ansys56\docu.



Because the “Open” action can only be associated with one setting, a user can easily generate multiple actions as shown in the middle dialog box above. This way, by right-clicking on the file, the user can select whether to use 128/32 MB for total workspace and database space or 256/100 MB instead. Different graphics drivers or different license products (Mechanical vs. Multiphysics) can also be distinguished in this manner.

In the above example, four actions are associated with the *.db file. If the user right-clicks on the ANSYS *.db file (as shown on right), these four commands appear. This gives the user more flexibility in running ANSYS with as little interaction as possible.

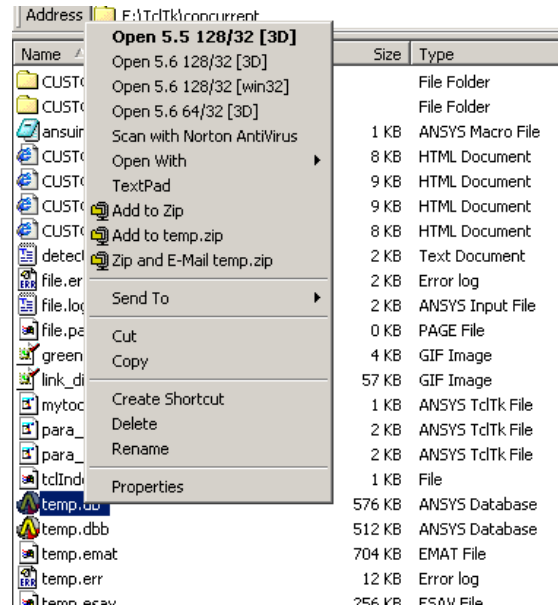
For the example on the right, three additional actions were created by selecting “New” in the “Edit File Type” dialog box above. Similar settings were used as before, although, this time, different graphics device and memory settings were input as follows:

```
Open 5.6 128/32 [3D]:
C:\..\launch_ansys56.bat "%1" 128 32 3D

Open 5.6 128/32 [win32]:
C:\..\launch_ansys56.bat "%1" 128 32 win32

Open 5.6 64/32 [3D]:
C:\..\launch_ansys56.bat "%1" 64 32 3D
```

An analogous DOS batch file can be created for ANSYS 5.4 or 5.5, giving the user flexibility in running older versions of ANSYS, if necessary. This is done by substituting the appropriate values of “56” with “54” or “55.” Because of this, the procedure outlined above is revision-independent. The author has used this procedure with success since ANSYS 5.4. Similarly, for companies with multiple types of licenses (Professional, Mechanical, Multiphysics, etc.), right-clicking the *.db file allows selection of the appropriate type of license, especially in the event that certain licenses such as Multiphysics are limited in quantity.



5. Discussion on Association of *.GRPH Display Graphics Files:

The ANSYS 5.6 file association using “displayw.exe -FILE %1” works quite nicely, so the user should not need to modify this technique for *.grph files. The use of this ANSYS-supplied association is recommended.

6. Discussion on Association of *.INP or *.DAT ANSYS Input Files:

ANSYS input files can also be associated with an action, similar to the *.db files above. Although the author will assume that ANSYS input files have the *.inp file extension, since these are plain text files, the user can have any file extension he/she wishes (*.txt, *.dat, etc.).

Usually, the default action (double-clicking the file) should be linked with a text editor, so the user can easily modify/read the input file simply by double-clicking on it. However, as noted in Section 4 above, multiple commands can be referenced to a file type. A user can add a command to automatically run the input file by right-clicking on it as follows:

1. Create a DOS batch file (regular text file) named “launch_ansys56.bat” as follows:

```
-----batch_ansys56.bat-----
@echo off
set PATH=%ANSYS56_DIR%\bin\%ANSYS_SYSDIR%;%PATH%
set ANS_CONSEC=YES
ansys -j %~n1 -m %2 -db %3 -p ane3flds -b -s read < "%~nx1" >> "%~n1.out"
-----end of file-----
```

Save this batch file in any location, such as C:\location_of_scripts.

2. Go in Windows Explorer, under “View > Folder Options” (or “Tools > Folder Options” in Windows 2000). Select the tab “File Types”. Scroll down to find the “INP” file extension (or create a new file type if it does not exist). Select “Edit” (or “Advanced” under Windows 2000). Select a “New” action, and type the following instead:



Action: Batch 5.6 128/32

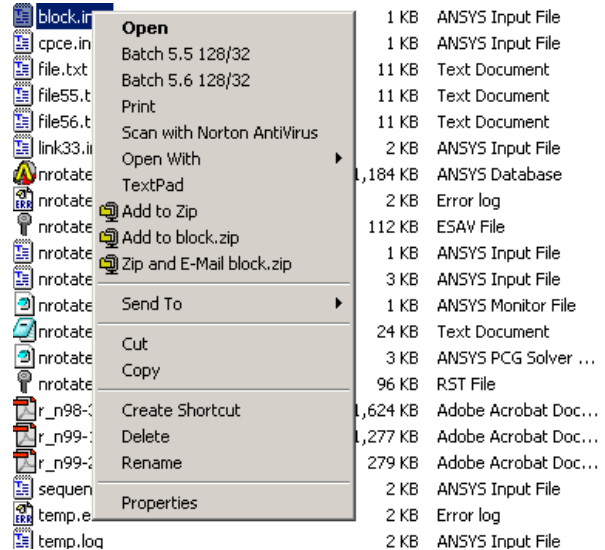
Application: C:\location_of_scripts\batch_ansys56.bat "%1" 128 32

This process adds an action if a user right-clicks on an ANSYS input file (*.inp) as shown on the right. By selecting "Batch 5.6 128/32" on the pop-up menu, ANSYS automatically runs the input file.

In Step 1 above, a DOS batch file is created which reads in an input file "%~nx1" (this passes file and extension to the DOS batch file). Output is redirected to a file called "jobname.out". The "-b" option tells ANSYS to run in batch mode. The product selection "-p" can be changed as noted in Section 3 above.

The ANS_CONSEC environment variable is only needed if the user does not wish to see the "Batch Job Completed" dialog box. If the user wants this notification, please omit the "set ANS_CONSEC=YES" line.

In Step 2, the command to run the input file in batch mode is added. Please note that this should *not* be set to the default action since a user will oftentimes want to have opening the text document the default action. Step 2 above will be a "New" action associated with the *.inp input file (in the example above, "Open" is the default action").



7. DOS Commands (Reference):

For the interested user, file association can be done in DOS with the "assoc" command, whereas file type definition is performed with the "ftype" command.

Sometimes, the *.inp (or other) text file is associated with notepad or some other text editor but this file type does not show up in the list of defined file types ("Edit > Folder Options > File Types"). This may be due to the fact that a description was not given to the filename at the time it was linked with notepad. Use the command

```
assoc .inp=
```

in a DOS window to delete that file association. Double-click on the *.inp file, and link it with notepad (or any text editor), but, this time, giving it a description. Now, the *.inp filetype should appear in Windows NT File Type dialog box.

8. Conclusion:

The use of DOS batch files makes linking ANSYS database files (*.db) and input files (*.inp) much easier in Windows Explorer. Not only can a user double-click on a *.db file to launch ANSYS and resume the database, but the user can configure the right-click pop-up menu to allow several other choices. This provides much flexibility while retaining the ease-of-use in Windows.

Some of this technique can be extended to UNIX, namely any platform running CDE (Common Desktop Environment) with the "dtfile" file manager. However, because most UNIX users are comfortable launching programs by typing in the xterm window, only the Windows method of file association was covered in this memo.

Sheldon Imaoka
Collaborative Solutions, Inc. (LA Office)
Engineering Consultant



ANSYS Tip of the Week

“ANSYS Tip of the Week” (TOTW) is provided for customers of Collaborative Solutions, Inc. (CSI) with active TECS agreements, distributed weekly in Adobe Acrobat PDF format via email. Unless otherwise stated, information contained herein should be applicable to ANSYS 5.4 and above, although usage of the latest version (5.6 as of this writing) is assumed. Users who wish to subscribe/unsubscribe or to view older TOTW archives can visit

http://www.csi-ansys.com/tip_of_the_week.htm

Corrections, comments, and suggestions are welcome and can be sent to operator@csi-ansys.com [they will be distributed to the appropriate person(s)]. While CSI engineers base their TOTW on technical support calls and user questions, ideas on future topics are appreciated. Users who wish to submit their own TOTW are encouraged to do so by emailing the above address for more information.

XANSYS Mailing List

The XANSYS mailing list is a forum for questions and discussions of the use of ANSYS. As of 04/00, there are more than 1000 subscribers with topics ranging from Structural, Thermal, Flotran, to Emag analyses, to name a few. Users are encouraged to subscribe to evaluate the usefulness of the mailing list for themselves. Also, either (a) using the mail program to filter [xansys] messages or (b) using the “digest” option to receive one combined email a day is strongly recommended to minimize sorting through the volume of postings.

This list is for *ALL* users of the ANSYS finite element analysis program from around the world. The list allows rapid communication among users concerning program bugs/ideas/modeling techniques. This list is NOT affiliated with ANSYS, Inc. even though several members of the ANSYS, Inc. staff are subscribers and regular contributors.

To SUBSCRIBE: send blank email to xansys-subscribe@onelist.com

To unsubscribe send blank email to xansys-unsubscribe@onelist.com

Archived on <http://www.infotech.tu-chemnitz.de/~messtech/ansys/ansys.html>

ANOTHER archive on <http://www.eScribe.com/software/xansys/>

(A poor archive is also at <http://www.onelist.com/archives.cgi/xansys>)

CSI ANSYS Technical Support, Training, & Mentoring

Collaborative Solutions, Inc. is committed to providing the best customer support in our industry. Three people will be devoted to technical support from 8:00 a.m. to 5:00 p.m. PST every working day. CSI customers with active TECS (maintenance) agreements may contact CSI by any of the following ways:

Phone: 760-431-4815

Fax: 760-431-4824

E-mail: firstname.lastname@csi-ansys.com

WWW: <http://www.csi-ansys.com>

FTP: <ftp://ftp.csi-ansys.com>

CSI Engineers:

Karen Dhuyvetter

Bill Bulat

Greg Miller

Sheldon Imaoka

Sean Harvey

David Haberman

Alfred Saad

Mike Rife

CSI believes strongly in the value of training and mentoring to help make customers successful using ANSYS. Training classes are usually 2-3 days in duration and provide instruction on various topics, including structural nonlinearities, heat transfer, and dynamics. Mentoring sessions involve working with a CSI engineer one-on-one on specific projects. These sessions help reinforce applicable subject matter covered in training classes or help ensure that the customer is using ANSYS most efficiently and effectively.

Training class schedules are posted at: <http://www.csi-ansys.com/training.htm>

Please contact your account manager for more details on training, mentoring, consulting, and other CSI services.