Collaborative Solutions Inc

An ANSYS Support Distributor

Date April 15, 2000 Memo Number STI49:000415A
Subject **ANSYS Tips & Tricks: Memory Management & Faster Solution Times**
Keywords Memory Management: /CONFIG: Optimal Solver Settings: Linear Static Structural

1. Introduction:

There are various configuration settings which may more efficiently utilize hardware resources for solution and postprocessing functions. This memo will cover a few of these options which may prove beneficial for users with (a) limited hardware and (b) performing linear static solutions at 5.6.

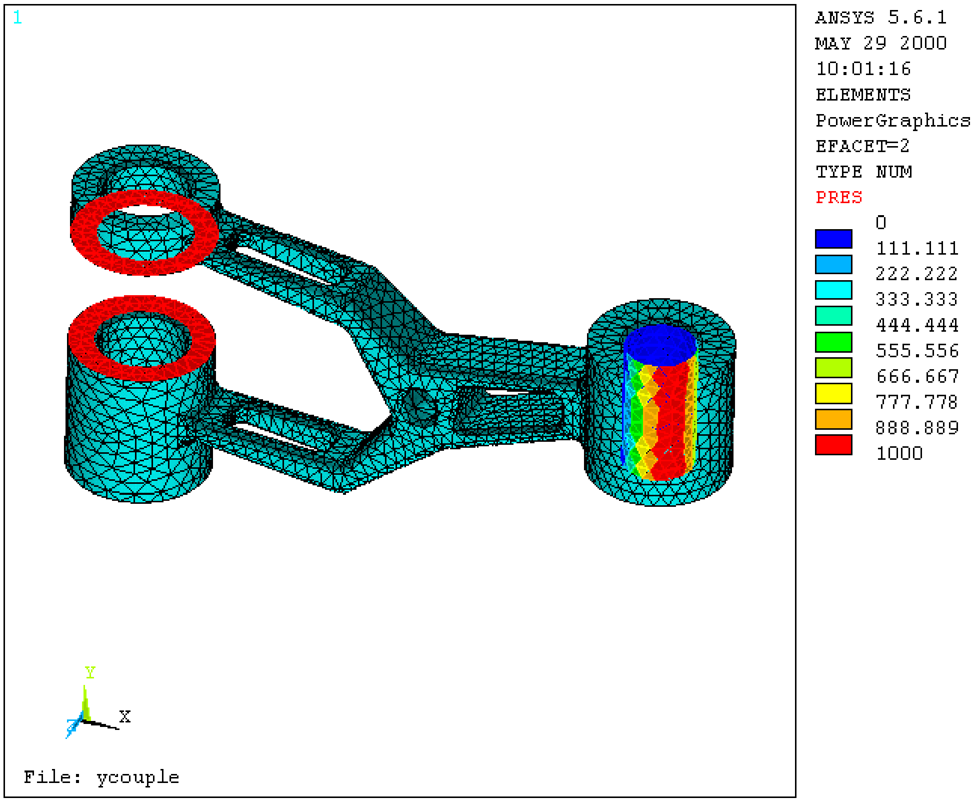
The author will use the performance of two computers as examples during this memo. However, this data should not be used for any direct comparisons of vendors' products or operating systems since one computer is a UNIX workstation whereas the other is a PC notebook, so one would expect the notebook to suffer in performance.

2. Description of Hardware and ANSYS Model Used in the Examples:

The author used two computers in this memo. A Dell Inspiron 7500 notebook running Windows NT 4.0 SP 6a with 500 MHz Intel processor and 256 MB of RAM with an internal IDE drive was used to demonstrate a system with limited hardware. This notebook's IDE drive is not as fast (read/write/seek times) compared with SCSI drives. This notebook also only has 256 MB of RAM.

Conversely, an HP C3000 running HP-UX 10.20 with a 400 MHz HPPA 8500 processor and 2 GB of RAM with two internal Ultrawide2 SCSI drive was used to demonstrate a system with good hardware. The Ultrawide2 SCSI drives have very fast read/write/seek times, and the 2GB of RAM provide an adequate amount of addressable memory.

For the various benchmarks, the model (with varying mesh density) shown below was used:



The model was constrained in many locations. Pressures were applied on the surfaces highlighted above. SOLID92 10-node tetrahedral elements were used, although some SURF154 surface effect elements were used for the parabolic pressure loading shown on the right-hand side of the model.

3. Use of Adequate Workspace and Database Memory:

The amount of "total workspace" memory in ANSYS specifies the amount of RAM (physical and virtual memory) that the ANSYS process will take. This can be done through the "-m" command-line option or in "Total Workspace" in the ANSYS Interactive Launcher.[1]

A portion of the "total workspace" is called "database" memory, which ANSYS will utilize for solid model geometry, finite element mesh, and results (basically, input and output). This database space can be thought of as the actual jobname.db file in RAM, so plotting, selection, and other manipulation can be done very quickly. The "database" space is specified either through the "-db" command-line option or in the "Database" field in the ANSYS Interactive Launcher.[2] If the "database space" is not large enough to contain the database (geometry) and 1 results set (if it exists), then the following note will appear:

```
*** NOTE ***                            CP=      38.120   TIME= 00:44:44
 Page file used.
```

When this happens, performance will degrade due to the page file created. A page file is ANSYS's own virtual memory scheme to store database (geometry + results) information to disk, in the event that everything does not fit into RAM. *It is always a good idea to avoid a page file as much as possible.* Exit out of ANSYS and specify more memory for the "database space", if possible. If the model is too large to fit in memory, a page file cannot be avoided.

The "scratch space" is the difference between the "total workspace" and the "database space". As the name implies, this is allocated memory used for temporary, internal calculations such as Boolean operations, element matrix formulation, etc.[3] Many portions of ANSYS support dynamic memory allocation, so if ANSYS requires more memory, it will try to "grab" any available memory from the operating system and produce the following note:

```
*** NOTE ***                            CP=      29.332   TIME= 11:42:30
 The initial memory allocation (-m) has been exceeded.
   Supplemental memory allocations are being used.
```

As a result, the "total workspace" (and, consequently, "scratch space") will grow, as needed.[4] For example, all solvers except the sparse solver (and the Block Lanczos eigenvalue extraction method) support dynamic memory allocation at 5.6. If an operation does not support dynamic memory allocation, the following message will be shown:

```
*** ERROR ***                           CP=      46.297   TIME= 11:47:06
 There is not enough memory for the Sparse Matrix Solver.
 Increase the memory using -m option (refer to the Basic Analysis
 Procedures Guide, Ch.  19).  Current memory available= 20 MB.
```

In the even that the above error message occurs, the user should save & exit out of ANSYS, specify a larger "total workspace" (i.e., larger "scratch space"), then perform the operation again.

When doing preprocessing or solution, one usually wants as big a scratch space as possible for Boolean operations, meshing, solving, etc. As a result, it is a good idea to specify a "database space" size a little larger than the actual jobname.db database file while specifying the "total workspace" to double this amount (or more, as needed).

On the other hand, when postprocessing, one usually wants a larger database space. This is due to the fact that, in memory, one has both the regular database and one results set for postprocessing. One can determine the size of the database+results set by performing a SAVE operation after solving. The size of the new jobname.db database file will be the combined size of the database geometry and results set; this amount should be specified as the "database space" whereas the "total workspace" can be up to double this amount. Not as much "scratch space" is needed during postprocessing.

At 5.7, the sparse solver (including the Block Lanczos eigenvalue extraction method which utilizes the sparse solver) supports dynamic memory allocation, so even if the total workspace requested is not large enough, ANSYS will grab additional available memory.

---

[1] For more details, please refer to Ch. 19.3.1 "Changing the Amount of ANSYS Work Space" in the ANSYS Basic Analysis Procedures Guide (BAS)

[2] For more details, please refer to Ch. 19.3.2 "Reallocating Database Space" in the BAS

[3] Binary file buffers are also stored in scratch space; this topic will not be covered in this present memo

[4] For more details, please refer to Ch. 19.2.2 "How ANSYS Uses its Work Space", last paragraph, in the BAS

Because of the conflict in the fact that preprocessing and solution want more scratch space whereas postprocessing requires more database space, herein lies a dilemma for users with limited hardware (for example, 128 to 256 MB of RAM and/or very slow hard drives, such as IDE drives).

The author prefers using the following technique for solving “large” problems on systems with limited amounts of RAM and/or slow hard drives:

1) Enter ANSYS with minimum database space needed and large enough total workspace
2) Perform preprocessing as usual, applying all loads and solution options, as required.
3) Exit ANSYS, saving “Geometry+Loads” only (i.e., essential information only)
4) Solve the problem in batch using input file discussed below
5) Enter ANSYS with larger database space (sum of database + one results set)
6) Perform postprocessing as usual

The input file mentioned at step (4) is shown below:

```
/batch                    ! Batch mode (not required)
resume                    ! Resume database called “jobname”
/config,norstgm,1         ! Do not store geometry in results file
/config,noeldbw,1         ! Do not write results to database
/solu                     ! Enter Solution processor
eqslv,pcg,1e-8            ! Specify a solver, if needed
solve                     ! Solve model
finish                    ! Finish Solution processor
/exit,nosave              ! Exit without saving the database
```

This input file assumes that the model being resumed is the same as the jobname (otherwise, the RESUME command can be modified accordingly). While this seems like a regular input file one may use to solve problems in batch mode, the important command is “/config,noeldb,1”.[5] Usually, ANSYS, during element calculations (e.g., stress recovery), copies the results to the database (“database space” in memory). The “/config,noeldb,1” command prevents this from occurring, and this is desirable for two reasons:

1) The results are written automatically to the jobname.rst file, so a copy of the results to the database is not required at this time.
2) It is assumed that this input file is used to solve in batch, not postprocess in batch. As a result, the user can specify the minimum database size (-db) for geometry information only, and keep the rest of RAM for solving purposes. Hence, with a minimum database size, during element calculation, if “/config,noeldb,1” was not used, ANSYS would want to write results to database, producing a pagefile which may slow performance.

A similar command, “/config,norstgm,1”, while not required, may be helpful for the user. It prevents geometric information of the FE mesh to be written to the results file. This may or may not be desired, depending on how the user does postprocessing. With the FE mesh information in the results file, one can postprocessing *without* first resuming the database. However, this results in a slightly larger results file since geometric information is written. As an example, for a static analysis of a 65,000 element database, the results file with geometric information (default behavior) is 84 MB. The same results with “/config,norstgm,1” (no geometric information written to results file) is 72 MB. For slow hard drives, this may result in small time savings and smaller results files.

---

[5] “/CONFIG,NOELDB,1” and “/CONFIG,NORSTGM,1” are documented in 5.6 but undocumented (yet available) at 5.5.

Table 1

🟦 Dell Inspiron 7500 Windows NT 4.0 SP 6a 500 MHz Intel 256 MB RAM (512 MB Virtual)
🟪 HP C3000 HP-UX 10.20 400 MHz PA8500 2 GB RAM
Comparison with /CONFIG, PCG vs AMG, MSAVE, and Tolerance

	Solver	RSTGM	ELDBW	DOF	Solve CPU	Solve Wall	%	Total CPU	Total Wall	%	-m	-db	Pagefile	Scratch
Case 1	PCG	0	0	106374	149	204	73%	181	321	56%	128	32	56.50	75.14
Case 2	PCG	1	1	106374	148	183	81%	178	254	70%	128	32	0.00	75.14
Case 1	PCG	0	0	105708	75	81	93%	103	115	89%	128	32	56.19	163.12
Case 2	PCG	1	1	105708	73	78	94%	100	109	91%	128	32	0.00	163.12
Case 11	PCG	0	0	207753	253	413	61%	314	844	37%	128	50	111.88	134.17
Case 12	PCG	1	1	207753	250	379	66%	295	484	61%	128	50	0.00	134.17
Case 11	PCG	0	0	206631	140	147	95%	192	201	96%	128	32	111.19	220.08
Case 12	PCG	1	1	206631	141	147	96%	184	193	95%	128	32	42.75	220.08
Case 21	PCG	0	0	301755	389	930	42%	480	1674	29%	128	64	163.25	194.53
Case 22	PCG	1	1	301755	387	856	45%	447	984	45%	128	64	0.00	194.53
Case 21	PCG	0	0	301386	217	226	96%	293	305	96%	128	32	163.00	281.21
Case 22	PCG	1	1	301386	208	218	96%	268	282	95%	128	32	61.63	281.21
Case 43	PCG	1	1	418461	266	280	95%	345	365	95%	128	32	84.63	357.95
Case 43c	PCG	1	1	418461	252	263	96%	329	343	96%	512	128	0.00	357.95
Case 53	PCG	1	1	505161	339	357	95%	433	456	95%	128	32	101.50	414.36
Case 53c	PCG	1	1	505161	308	324	95%	400	419	96%	512	128	0.00	414.36

For the example model presented in Section 2 earlier, the results are summarized in Table 1 for the Dell Inspiron 7500 and HP C3000.

The Dell Inspiron 7500 results are shown in light blue. Note that because of the IDE drives, the CPU time is anywhere from 30-70% of total elapsed time. This is an indication that the hardware is slow. If one compares the results for Case 1 and Case 2 (both in blue), it is quite clear that the "-db" size of 32 MB is too small to fit both geometry/input and results/output. Consequently, during element calculation, a page file of 57 MB is created. This slows down the total time of Case 1, so Case 1 takes about a minute slower than Case 2. This difference is exaggerated for Case 21 and 22 where Case 22 takes 16.4 minutes whereas Case 21 takes 27.9 minutes.

On the other hand, the HP C3000 shows little difference with the creation of a page file. Note that the CPU time is usually 90% or greater of total elapsed time, indicating that the Ultrawide2 SCSI drives have very fast write times. This shows that the hardware is so fast that changing options such as "/CONFIG,NOELDB,1" provide little benefit, and the results reflect this assumption. In the worst case of a 500,000 DOF model (Cases 53 and 53c), a page file of 101.50 MB is generated for Case 53, yet the final solution time is only 30 seconds slower than Case 53c.

As noted earlier, this technique is good for users with slow drives or limited physical RAM; for those individuals, this may provide some time savings (consider cases of nonlinear or transient analyses which would result in greater time savings). For users with fast hard drives and/or enough RAM, this extra effort may not be worthwhile.

4. Use of MSAVE for PCG Solver

Whenever possible, the user should always use the PCG solver for linear structural analyses containing SOLID elements since the PCG solver is much faster than the frontal or sparse solvers.[6]

The MSAVE option for the PCG solver is new at 5.6. The PCG Solver usually is quite memory-intensive, but the MSAVE option is used to reduce these memory requirements by performing an element-by-element solution approach.

MSAVE currently can only be used under the following conditions:

- PCG Solver only
- Used for parts of the model comprised of SOLID92 elements
- Linear analysis (e.g., small strains & deflections, linear material behavior)
- Static or full transient analyses

For most linear static analyses of CAD-imported geometry, these criteria are fulfilled, so a user can take advantage of the MSAVE command, if needed. This, like the technique outlined in Section 3, is commonly used in cases with limited hardware (not enough RAM).

It is instructive to note that this option should not be used all the time. The element-by-element approach takes more CPU time than a regular PCG solution as shown in Table 2 below:

Table 2

🟦 Dell Inspiron 7500 Windows NT 4.0 SP 6a 500 MHz Intel 256 MB RAM (512 MB Virtual)
🟪 HP C3000 HP-UX 10.20 400 MHz PA8500 2 GB RAM

Comparison with /CONFIG, PCG vs AMG, MSAVE, and Tolerance

	Solver	Tolerance	Esize	Msave	DOF	Solve CPU	Solve Wall	%	Total CPU	Total Wall	%	-m	-db	Pagefile	Scratch
Case 12	PCG	1.00E-08	0.15	off	207753	250	379	66%	295	484	61%	128	50	0.00	134.17
Case 14	PCG	1.00E-08	0.15	on	207753	397	452	88%	443	544	81%	128	50	0.00	117.26
Case 12	PCG	1.00E-08	0.15	off	206631	141	147	96%	184	193	95%	128	32	42.75	220.08
Case 14	PCG	1.00E-08	0.15	on	206631	231	236	98%	274	282	97%	128	32	42.75	173.99
Case 22	PCG	1.00E-08	0.125	off	301755	387	856	45%	447	984	45%	128	64	0.00	194.53
Case 24	PCG	1.00E-08	0.125	on	301755	612	721	85%	672	840	80%	128	64	0.00	177.62
Case 22	PCG	1.00E-08	0.125	off	301386	208	218	96%	268	282	95%	128	32	61.63	281.21
Case 24	PCG	1.00E-08	0.125	on	301386	353	360	98%	413	424	97%	128	32	61.63	184.21
Case 33	PCG	1.00E-06	0.115	off	363912	347	1305	27%	416	1439	29%	192	90	0.00	235.38
Case 37	PCG	1.00E-06	0.115	on	363912	521	742	70%	590	876	67%	192	90	0.00	215.88
Case 33	PCG	1.00E-06	0.115	off	365046	222	233	95%	293	308	95%	128	32	74.13	322.74
Case 37	PCG	1.00E-06	0.115	on	365046	342	349	98%	412	424	97%	128	32	74.13	216.37
Case 43	PCG	1.00E-06	0.11	off	417882	425	1867	23%	502	2009	25%	192	90	0.00	270.94
Case 47	PCG	1.00E-06	0.11	on	417882	637	881	72%	715	1026	70%	192	90	0.00	243.98
Case 43c	PCG	1.00E-06	0.11	off	418461	252	263	96%	329	343	96%	512	128	0.00	357.95
Case 47c	PCG	1.00E-06	0.11	on	418461	392	397	99%	468	477	98%	512	128	0.00	243.98
Case 53c	PCG	1.00E-06	0.1	off	505161	308	324	95%	400	419	96%	512	128	0.00	414.36
Case 57c	PCG	1.00E-06	0.1	on	505161	480	487	98%	571	581	98%	512	128	0.00	300.80

Note that, as shown in Table 2 above, the CPU time required for the MSAVE option is always much larger than a normal solution. This is to be expected since this is an element-by-element assembly rather than an assemblage of the global stiffness matrix.

Recall that the HP C3000 workstation used has 2 GB of RAM. As a result, the MSAVE option provides no benefit (actually, MSAVE results in longer run times) up to 500,000 DOF models.

On the other hand, MSAVE provides faster solution times for the Dell Inspiron 7500 notebook, *when the amount of RAM is surpassed*. Note that in Cases 12 and 14, the scratch space is ~130 MB, so this still fits in RAM (256 MB of physical RAM).[7] Hence, Case 12 (MSAVE,OFF) is faster than Case 14 (MSAVE,ON). Once the model size gets larger than can be fit in RAM, as in the examples of Case 33 and 37 where the scratch space required is ~235 MB, MSAVE provides faster runtimes. In fact, in Cases 43 and 47 of 400,000 DOF models, the total elapsed time with MSAVE,ON is 50% that of the regular PCG solver. The benefit MSAVE provides is limited, however, so a user cannot run *any* large model on limited hardware; for example, a 500,000 DOF model was run on the Inspiron 7500, but this was too large of a model to solve in any reasonable amount of time (< 3 hours), even with MSAVE,ON.

---

[6] For BEAM and SHELL elements, the user may find that the sparse solver provides faster solution times than the PCG solver. Solver selection is discussed in more detail in the ANSYS manuals as well as in CSI's training courses.

[7] The scratch space used (with dynamic memory allocation) is printed at the end of an ANSYS session. This information is written to the end of the output file in batch mode (as well as a UNIX interactive session if run from the xansys56/tansys56 launcher).

5. Limiting Tolerance for PCG (and Other Iterative) Solvers:

The conjugate gradient solvers in ANSYS "guess" a solution vector and update this guess at every solver iteration. Hence, in the case of a static analysis:

$$[K]\{u\} = \{F\}$$

the iterative (conjugate gradient) solvers start off with a guess (of zero) of {u}. Then, the solver iterates, updating its guess of the solution vector {u}. Convergence is achieved when:

$$\frac{\{R_i\}^T\{R_i\}}{\{F\}^T\{F\}} \le \varepsilon^2$$

where $\{R_i\}$ is the residual vector $\{R_i\} = \{F\} - [K]\{u_i\}$.[8]

The tolerance level ε is supplied via the second argument of the EQSLV command. For static analyses, the default value of tolerance is 1e-8, which provides high accuracy. The author has found that tolerance level of 1e-6 is quite sufficient for linear static analyses. This, too, would help speed up analyses as shown in Table 3 below:

Table 3

🟦	Dell Inspiron 7500 Windows NT 4.0 SP 6a 500 MHz Intel 256 MB RAM (512 MB Virtual)
🟪	HP C3000 HP-UX 10.20 400 MHz PA8500 2 GB RAM

Comparison with /CONFIG, PCG vs AMG, MSAVE, and Tolerance

	Solver	Tolerance	Esize	DOF	Solve CPU	Solve Wall	%	Total CPU	Total Wall	%
Case 2	PCG	1.00E-08	0.2	106374	148	183	81%	178	254	70%
Case 3	PCG	1.00E-06	0.2	106374	119	152	78%	148	226	66%
Case 2	PCG	1.00E-08	0.2	105708	73	78	94%	100	109	91%
Case 3	PCG	1.00E-06	0.2	105708	65	69	94%	91	100	91%
Case 12	PCG	1.00E-08	0.15	207753	250	379	66%	295	484	61%
Case 13	PCG	1.00E-06	0.15	207753	205	337	61%	251	434	58%
Case 12	PCG	1.00E-08	0.15	206631	141	147	96%	184	193	95%
Case 13	PCG	1.00E-06	0.15	206631	122	128	95%	165	173	95%
Case 22	PCG	1.00E-08	0.125	301755	387	856	45%	447	984	45%
Case 23	PCG	1.00E-06	0.125	301755	309	792	39%	370	914	40%
Case 22	PCG	1.00E-08	0.125	301386	208	218	96%	268	282	95%
Case 23	PCG	1.00E-06	0.125	301386	188	197	96%	248	260	95%

As shown above, for both the Inspiron 7500 (in light blue) and the C3000 (in purple), lowering the tolerance for the PCG solver will speed up solution and overall times.

For fast processors (e.g., HPPA8500 400 MHz processor, purple), the time savings are minimal (e.g, 260 vs. 282 seconds for 300,000 DOF model, Cases 22 and 23).

For "regular" processors (e.g., Intel 500 MHz processor, light blue), the time savings are somewhat more noticable, about 10% reduction time in speed.

One can perform benchmarks for his/her own purposes, but one should find that dropping the tolerance level from 1e-8 to 1e-6 results in unnoticeable change in results and accuracy. However, the author does not recommend lowering the PCG solver tolerance less than 1e-6, *especially* for transient and nonlinear analyses, where the buildup of error due to a loose/large tolerance specification may produce inaccurate results.

At 5.5, there is a new command PRECISION, which changes the accuracy of the PCG solver from double precision to single precision. While this may result in memory savings, the author does *not* recommend using the PRECISION command, *especially* in conjunction with lowering the tolerance of the PCG solver (defaults is 1e-5 for PRECISION,SINGLE) since he has found that erroneous results may occur due to numerical precision (single precision accuracy stores less significant digits).

---

[8] For more information, please refer to Ch. 15.13 "Conjugate Gradient Solvers" in the ANSYS Theory Manual.

6. Direct Assembly of Equations:

For linear static and full transient structural analyses containing SOLID92, direct assembly of equations is performed, and no EMAT or ESAV files will be created.[9] This involves creating the global stiffness matrix without needing to first form the element stiffness matrices. Other elements such as PLANE2, SOLID45, and SOLID95 also support direct assembly of equations. Note that the existence of certain elements such as surface effect elements will still allow direct assemblage of matrices.

When these conditions have been met, the following note will be printed during solution:

```
*** NOTE ***                            CP=      30.083   TIME= 23:11:05
 The conditions for direct assembly have been met.  No .emat or .erot
  files will be produced.
```

For machines with fast disk I/O, the creation of element matrices will not affect total elapsed time considerably. However, for the other cases, this may impact total solution time as noted below:

Table 4

🟦	Dell Inspiron 7500 Windows NT 4.0 SP 6a 500 MHz Intel 256 MB RAM (512 MB Virtual)
🟪	HP C3000 HP-UX 10.20 400 MHz PA8500 2 GB RAM

Comparison with /CONFIG, PCG vs AMG, MSAVE, and Tolerance

	Solver	Tolerance	Esize	Direct	DOF	Solve CPU	Solve Wall	%	Total CPU	Total Wall	%
Case 33	PCG	1.00E-06	0.115	on	363912	347	1305	27%	416	1439	29%
Case 33b	PCG	1.00E-06	0.115	off	363912	412	1716	24%	481	1852	26%
Case 37	PCG	1.00E-06	0.115	on	363912	521	742	70%	590	876	67%
Case 37b	PCG	1.00E-06	0.115	off	363912	638	1702	37%	708	1842	38%
Case 43	PCG	1.00E-06	0.11	on	417882	425	1867	23%	502	2009	25%
Case 43b	PCG	1.00E-06	0.11	off	417882	492	2171	23%	571	2319	25%
Case 47	PCG	1.00E-06	0.11	on	417882	637	881	72%	715	1026	70%
Case 47b	PCG	1.00E-06	0.11	off	417882	769	1981	39%	847	2137	40%

The column "Direct" notes whether direct assembly of matrices has been used. As noted in the case of the Inspiron 7500 above, the use of direct assembly of equations results in much faster solution times. Cases 37 and 47 use MSAVE,ON (element-by-element assembly) as well, and this is where the most pronounced change can be seen.

The user cannot control direct assembly of equations, so there are no options and such to specify. The only thing that the user can do is to use the above-noted elements, if possible, for linear static analyses, so ANSYS can perform direct assembly of equations automatically. If the conditions have been met, the above NOTE will appear in the beginning of solution in the output window/file.

At 5.7,use of direct assembly of equations has been extended to all structural, thermal, and electrostatic elements.

7. Summary:

This memo described some options the user can implement on limited hardware to ensure optimal performance. These settings involved:

- Proper use of total workspace (-m) and database space (-db) settings
- Use of the PCG solver, including MSAVE and tolerance control
- Automatic direct assembly of equations

There are many other options not covered in this memo, but this memo hopes to introduce some of the key points to the user. CSI customers may also call CSI's tech support line for help in any configuration of settings as well. Future memos may hope to cover recommended settings for nonlinear and dynamic analyses.

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

---

[9] The same applies for all thermal elements in both linear and nonlinear, steady-state and transient analyses.

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.