

file:///C:/Program%20Files/ANSYS%20Inc/v202/aisol/DesignSpace/DSPages/html/DSS...

file:///C:/Program%20Fi

file:///C:/Program%20Files/ANSYS%20Inc/v2

file:///C:/Program%20Files/ANSYS%20Inc

file://

file:///C:/Program%20File

file:///C:/Program%20Files/ANSYS%2

file:///C:/Program%20Files/ANSYS

```

TOTAL OF      2 CORES REQUESTED
INPUT FILE NAME      = D:\_LOCAL_prgWS\AdvanceNumericalMethod\_ProjectScratch\Scr5937\dummy.dat
OUTPUT FILE NAME     = D:\_LOCAL_prgWS\AdvanceNumericalMethod\_ProjectScratch\Scr5937\solve.out
START-UP FILE MODE   = NOREAD
STOP FILE MODE       = NOREAD

RELEASE= 2020 R2      BUILD= 20.2      UP20200601  VERSION=WINDOWS x64
CURRENT JOBNAME=file  19:50:38  MAY 24, 2025 CP=      0.406

PARAMETER _DS_PROGRESS =      999.0000000

/INPUT FILE= ds.dat  LINE=      0

DO NOT WRITE ELEMENT RESULTS INTO DATABASE

*GET _WALLSTRT FROM ACTI  ITEM=TIME WALL  VALUE=  19.8438889

TITLE=
WB_femKetCauPhangBtl--Static Structural (D4)

SET PARAMETER DIMENSIONS ON _WB_PROJECTSCRATCH_DIR
TYPE=STRI  DIMENSIONS=      248      1      1

PARAMETER _WB_PROJECTSCRATCH_DIR(1) = D:\_LOCAL_prgWS\AdvanceNumericalMethod\_ProjectScratch\Scr5937\

SET PARAMETER DIMENSIONS ON _WB_SOLVERFILES_DIR
TYPE=STRI  DIMENSIONS=      248      1      1

PARAMETER _WB_SOLVERFILES_DIR(1) = D:\_LOCAL_prgWS\AdvanceNumericalMethod\WB_femKetCauPhangBtl_files\dp0\SYS-4\MECH\

SET PARAMETER DIMENSIONS ON _WB_USERFILES_DIR
TYPE=STRI  DIMENSIONS=      248      1      1

PARAMETER _WB_USERFILES_DIR(1) = D:\_LOCAL_prgWS\AdvanceNumericalMethod\WB_femKetCauPhangBtl_files\user_files\
--- Data in consistent MKS units. See Solving Units in the help system for more

MKS UNITS SPECIFIED FOR INTERNAL
LENGTH      (l) = METER (M)
MASS        (M) = KILOGRAM (KG)
TIME        (t) = SECOND (SEC)
TEMPERATURE (T) = CELSIUS (C)
TOFFSET     = 273.0
CHARGE      (Q) = COULOMB
FORCE       (f) = NEWTON (N) (KG-M/SEC2)
HEAT        = JOULE (N-M)

PRESSURE     = PASCAL (NEWTON/M**2)
ENERGY       (W) = JOULE (N-M)
POWER        (P) = WATT (N-M/SEC)
CURRENT      (i) = AMPERE (COULOMBS/SEC)
CAPACITANCE (C) = FARAD
INDUCTANCE   (L) = HENRY
MAGNETIC FLUX = WEBER
RESISTANCE   (R) = OHM
ELECTRIC POTENTIAL = VOLT

INPUT UNITS ARE ALSO SET TO MKS

*** ANSYS - ENGINEERING ANALYSIS SYSTEM  RELEASE 2020 R2      20.2      ***
ANSYS Mechanical Enterprise
20120530  VERSION=WINDOWS x64  19:50:38  MAY 24, 2025 CP=      0.484

WB_femKetCauPhangBtl--Static Structural (D4)

***** ANSYS ANALYSIS DEFINITION (PREP7) *****
***** Nodes for the whole assembly *****
***** Elements for Body 1 "Surface Body" *****
***** Elements for Body 2 "Surface Body" *****
***** Elements for Body 3 "Surface Body" *****
***** Elements for Body 4 "Surface Body" *****
***** Send User Defined Coordinate System(s) *****
***** Set Reference Temperature *****
***** Send Materials *****
***** Send Sheet Properties *****
***** Fixed Supports *****
***** Define Pressure Vector Using Surface Effect Elements *****
***** Define Pressure Vector Using Surface Effect Elements *****
***** Define Vertex Force Using Follower Elements *****

**** ROUTINE COMPLETED **** CP =      0.484

File compression level set to 0 for RST      file(s).
--- Number of total nodes = 10
--- Number of contact elements = 3
--- Number of spring elements = 0
--- Number of bearing elements = 0
--- Number of solid elements = 4
--- Number of condensed parts = 0
--- Number of total elements = 9

*GET _WALLBSOL FROM ACTI  ITEM=TIME WALL  VALUE=  19.8438889
*****
***** SOLUTION *****
*****

**** ANSYS SOLUTION ROUTINE ****

PERFORM A STATIC ANALYSIS
THIS WILL BE A NEW ANALYSIS

PARAMETER _THICKRATIO =      0.000000000

```

```

USE SPARSE MATRIX DIRECT SOLVER

CONTACT INFORMATION PRINTOUT LEVEL      1

NLDIAG: Nonlinear diagnostics CONT option is set to ON.
      Writing frequency : each ITERATION.

DEFINE RESTART CONTROL FOR LOADSTEP LAST
AT FREQUENCY OF LAST AND NUMBER FOR OVERWRITE IS  -1

WRITE MULTIFRAME RESTART FILES EVEN
WHEN IT IS A LINEAR STATIC ANALYSIS
*****
***** SOLVE FOR LS 1 OF 1 *****

SELECT      FOR ITEM=TYPE COMPONENT=
IN RANGE    5 TO      5 STEP      1

      2 ELEMENTS (OF      9 DEFINED) SELECTED BY ESEL COMMAND.

SELECT      ALL NODES HAVING ANY ELEMENT IN ELEMENT SET.

      3 NODES (OF      10 DEFINED) SELECTED FROM
      2 SELECTED ELEMENTS BY NSLE COMMAND.

SPECIFIED SURFACE LOAD PRES FOR ALL SELECTED ELEMENTS LKEY = 1 KVAL = 1
VALUES =      0.0000      0.0000      0.0000      0.0000

SPECIFIED SURFACE LOAD PRES FOR ALL SELECTED ELEMENTS LKEY = 2 KVAL = 1
VALUES = -0.30000E+06 -0.30000E+06 -0.30000E+06 -0.30000E+06

SELECT      FOR ITEM=TYPE COMPONENT=
IN RANGE    6 TO      6 STEP      1

      1 ELEMENTS (OF      9 DEFINED) SELECTED BY ESEL COMMAND.

SELECT      ALL NODES HAVING ANY ELEMENT IN ELEMENT SET.

      2 NODES (OF      10 DEFINED) SELECTED FROM
      1 SELECTED ELEMENTS BY NSLE COMMAND.

SPECIFIED SURFACE LOAD PRES FOR ALL SELECTED ELEMENTS LKEY = 1 KVAL = 1
VALUES =      0.15000E+06      0.15000E+06      0.15000E+06      0.15000E+06

SPECIFIED SURFACE LOAD PRES FOR ALL SELECTED ELEMENTS LKEY = 2 KVAL = 1
VALUES =      0.0000      0.0000      0.0000      0.0000

ALL SELECT  FOR ITEM=NODE COMPONENT=
IN RANGE    1 TO      10 STEP      1

      10 NODES (OF      10 DEFINED) SELECTED BY NSEL COMMAND.

ALL SELECT  FOR ITEM=ELEM COMPONENT=
IN RANGE    1 TO      9 STEP      1

      9 ELEMENTS (OF      9 DEFINED) SELECTED BY ESEL COMMAND.

SELECT      FOR ITEM=REAL COMPONENT=
IN RANGE    7 TO      7 STEP      1

      1 ELEMENTS (OF      9 DEFINED) SELECTED BY ESEL COMMAND.

SPECIFIED SURFACE LOAD PRES FOR ALL SELECTED ELEMENTS LKEY = 1 KVAL = 1
VALUES =      0.10000E+06      0.10000E+06      0.10000E+06      0.10000E+06

SELECT      FOR ITEM=REAL COMPONENT=
IN RANGE    8 TO      8 STEP      1

      1 ELEMENTS (OF      9 DEFINED) SELECTED BY ESEL COMMAND.

SPECIFIED SURFACE LOAD PRES FOR ALL SELECTED ELEMENTS LKEY = 1 KVAL = 1
VALUES = -0.17321E+06 -0.17321E+06 -0.17321E+06 -0.17321E+06

ALL SELECT  FOR ITEM=ELEM COMPONENT=
IN RANGE    1 TO      9 STEP      1

      9 ELEMENTS (OF      9 DEFINED) SELECTED BY ESEL COMMAND.

PRINTOUT RESUMED BY /GOP

USE      1 SUBSTEPS INITIALLY THIS LOAD STEP FOR ALL DEGREES OF FREEDOM
FOR AUTOMATIC TIME STEPPING:
USE      1 SUBSTEPS AS A MAXIMUM
USE      1 SUBSTEPS AS A MINIMUM

TIME= 1.0000

ERASE THE CURRENT DATABASE OUTPUT CONTROL TABLE.

WRITE ALL ITEMS TO THE DATABASE WITH A FREQUENCY OF NONE
FOR ALL APPLICABLE ENTITIES

WRITE NSOL ITEMS TO THE DATABASE WITH A FREQUENCY OF ALL
FOR ALL APPLICABLE ENTITIES

WRITE RSOL ITEMS TO THE DATABASE WITH A FREQUENCY OF ALL
FOR ALL APPLICABLE ENTITIES

WRITE EANG ITEMS TO THE DATABASE WITH A FREQUENCY OF ALL
FOR ALL APPLICABLE ENTITIES

WRITE ETMP ITEMS TO THE DATABASE WITH A FREQUENCY OF ALL
FOR ALL APPLICABLE ENTITIES

WRITE VENG ITEMS TO THE DATABASE WITH A FREQUENCY OF ALL
FOR ALL APPLICABLE ENTITIES

WRITE NLOA ITEMS TO THE DATABASE WITH A FREQUENCY OF ALL
FOR ALL APPLICABLE ENTITIES

```

```

WRITE STRS ITEMS TO THE DATABASE WITH A FREQUENCY OF ALL
FOR ALL APPLICABLE ENTITIES

WRITE EPEL ITEMS TO THE DATABASE WITH A FREQUENCY OF ALL
FOR ALL APPLICABLE ENTITIES

WRITE EPPL ITEMS TO THE DATABASE WITH A FREQUENCY OF ALL
FOR ALL APPLICABLE ENTITIES

WRITE CONT ITEMS TO THE DATABASE WITH A FREQUENCY OF ALL
FOR ALL APPLICABLE ENTITIES

FINISH SOLUTION PROCESSING

***** ROUTINE COMPLETED ***** CP = 0.484

*** ANSYS - ENGINEERING ANALYSIS SYSTEM RELEASE 2020 R2 20.2 ***
ANSYS Mechanical Enterprise
20120530 VERSION=WINDOWS x64 19:50:38 MAY 24, 2025 CP= 0.484
WB_femKetCauPhangBtl--Static Structural (D4)

***** ANSYS ANALYSIS DEFINITION (PREP7) *****

*DO LOOP ON PARAMETER= I FROM 1.0000 TO 4.0000 BY 1.0000

ELEMENT TYPE 1 IS PLANE182 2-D 4-NODE PLANE STRS SOLID
KEYOPT( 1- 6)= 0 0 0 0 0 0
KEYOPT( 7-12)= 0 0 0 0 0 0
KEYOPT(13-18)= 0 0 0 0 0 0

CURRENT NODAL DOF SET IS UX UY UZ
THREE-DIMENSIONAL MODEL

ELEMENT TYPE 1 IS PLANE182 2-D 4-NODE PLANE STRS SOLID
KEYOPT( 1- 6)= 0 0 0 0 0 0
KEYOPT( 7-12)= 0 0 0 0 0 0
KEYOPT(13-18)= 0 0 0 0 0 0

CURRENT NODAL DOF SET IS UX UY UZ
THREE-DIMENSIONAL MODEL

ELEMENT TYPE 1 IS PLANE182 2-D 4-NODE PLANE STRS SOLID
KEYOPT( 1- 6)= 0 0 0 0 0 0
KEYOPT( 7-12)= 0 0 0 0 0 0
KEYOPT(13-18)= 0 0 0 0 0 0

CURRENT NODAL DOF SET IS UX UY UZ
THREE-DIMENSIONAL MODEL

ELEMENT TYPE 1 IS PLANE182 2-D 4-NODE PLANE STRN SOLID
KEYOPT( 1- 6)= 0 0 2 0 0 0
KEYOPT( 7-12)= 0 0 0 0 0 0
KEYOPT(13-18)= 0 0 0 0 0 0

CURRENT NODAL DOF SET IS UX UY UZ
THREE-DIMENSIONAL MODEL

*ENDDO INDEX= I

***** ROUTINE COMPLETED ***** CP = 0.484

***** ANSYS SOLUTION ROUTINE *****

FORCE WRITING OF THE EMAT FILE

EXTRAPOLATE ALL APPROPRIATE INTEGRATION POINT VALUES TO THE NODES

*GET ANSINTER_ FROM ACT1 ITEM=INT VALUE= 0.00000000

*IF ANSINTER_ ( = 0.00000 ) NE
0 ( = 0.00000 ) THEN

*ENDIF

***** ANSYS SOLVE COMMAND *****

*** WARNING *** CP = 0.484 TIME= 19:50:38
Element shape checking is currently inactive. Issue SHPP,ON or
SHPP,WARN to reactivate, if desired.

*** NOTE *** CP = 0.484 TIME= 19:50:38
The model data was checked and warning messages were found.
Please review output or errors file (
D:\_LOCAL_prjWS\AdvanceNumericalMethod\_ProjectScratch\Scr5937\file.err
r ) for these warning messages.

*** SELECTION OF ELEMENT TECHNOLOGIES FOR APPLICABLE ELEMENTS ***
--- GIVE SUGGESTIONS AND RESET THE KEY OPTIONS ---

ELEMENT TYPE 1 IS PLANE182 WITH PLANE STRAIN OPTION. IT IS ASSOCIATED WITH
LINEAR MATERIALS ONLY AND POISSON'S RATIO IS NOT GREATER THAN 0.49. KEYOPT(1)=3
IS SUGGESTED AND HAS BEEN RESET.
KEYOPT(1-12)= 3 0 2 0 0 0 0 0 0 0 0 0

ELEMENT TYPE 2 IS PLANE182 WITH PLANE STRAIN OPTION. IT IS ASSOCIATED WITH
LINEAR MATERIALS ONLY AND POISSON'S RATIO IS NOT GREATER THAN 0.49. KEYOPT(1)=3
IS SUGGESTED AND HAS BEEN RESET.
KEYOPT(1-12)= 3 0 2 0 0 0 0 0 0 0 0 0

ELEMENT TYPE 3 IS PLANE182 WITH PLANE STRAIN OPTION. IT IS ASSOCIATED WITH
LINEAR MATERIALS ONLY AND POISSON'S RATIO IS NOT GREATER THAN 0.49. KEYOPT(1)=3

```

IS SUGGESTED AND HAS BEEN RESET.  
 KEYOPT(1-12)= 3 0 2 0 0 0 0 0 0 0 0 0  
 ELEMENT TYPE 4 IS PLANE182 WITH PLANE STRAIN OPTION. IT IS ASSOCIATED WITH  
 LINEAR MATERIALS ONLY AND POISSON'S RATIO IS NOT GREATER THAN 0.49. KEYOPT(1)=3  
 IS SUGGESTED AND HAS BEEN RESET.  
 KEYOPT(1-12)= 3 0 2 0 0 0 0 0 0 0 0 0

\*\*\* ANSYS - ENGINEERING ANALYSIS SYSTEM RELEASE 2020 R2 20.2 \*\*\*  
 ANSYS Mechanical Enterprise  
 20120530 VERSION=WINDOWS x64 19:50:38 MAY 24, 2025 CP= 0.484

WB\_femKetCauPhangBtl--Static Structural (D4)

#### SOLUTION OPTIONS

PROBLEM DIMENSIONALITY. . . . .3-D  
 DEGREES OF FREEDOM. . . . . UX UY UZ  
 ANALYSIS TYPE. . . . .STATIC (STEADY-STATE)  
 OFFSET TEMPERATURE FROM ABSOLUTE ZERO. . . . .273.15  
 EQUATION SOLVER OPTION. . . . .SPARSE  
 GLOBALLY ASSEMBLED MATRIX. . . . .SYMMETRIC

\*\*\* WARNING \*\*\* CP = 0.484 TIME= 19:50:38  
 Material number 5 (used by element 5) should normally have at least one  
 MP or one TB type command associated with it. Output of energy by  
 material may not be available.

\*\*\* NOTE \*\*\* CP = 0.484 TIME= 19:50:38  
 The step data was checked and warning messages were found.  
 Please review output or errors file (  
 D:\\_LOCAL\_prgWS\AdvanceNumericalMethod\\_ProjectScratch\Scr5937\file.err  
 r ) for these warning messages.

#### LOAD STEP OPTIONS

LOAD STEP NUMBER. . . . .1  
 TIME AT END OF THE LOAD STEP. . . . .1.0000  
 NUMBER OF SUBSTEPS. . . . .1  
 STEP CHANGE BOUNDARY CONDITIONS. . . . .NO  
 EXTRAPOLATE INTEGRATION POINT VALUES TO NODE. . . . .YES  
 PRINT OUTPUT CONTROLS. . . . .NO PRINTOUT  
 DATABASE OUTPUT CONTROLS

ITEM	FREQUENCY	COMPONENT
ALL	NONE	
NSOL	ALL	
RSOL	ALL	
EANG	ALL	
ETMP	ALL	
VENG	ALL	
NLOA	ALL	
STRS	ALL	
EPEL	ALL	
EPPL	ALL	
CONT	ALL	

SOLUTION MONITORING INFO IS WRITTEN TO FILE=  
 file.mntr

\*\*\*\* CENTER OF MASS, MASS, AND MASS MOMENTS OF INERTIA \*\*\*\*

CALCULATIONS ASSUME ELEMENT MASS AT ELEMENT CENTROID

TOTAL MASS = 1879.2

CENTER OF MASS	MOM. OF INERTIA ABOUT ORIGIN	MOM. OF INERTIA ABOUT CENTER OF MASS
XC = 0.52500	IXX = 131.5	IXX = 14.09
YC = 0.25000	IYY = 634.2	IYY = 116.3
ZC = 0.0000	IZZ = 765.8	IZZ = 130.4
	IXY = -225.5	IXY = 21.14
	IYZ = 0.000	IYZ = 0.000
	IZX = 0.000	IZX = 0.000

\*\*\* MASS SUMMARY BY ELEMENT TYPE \*\*\*

TYPE	MASS
1	469.800
2	469.800
3	469.800
4	469.800

Range of element maximum matrix coefficients in global coordinates  
 Maximum = 1.224254261E+11 at element 3.  
 Minimum = 1.224254261E+11 at element 2.

\*\*\* ELEMENT MATRIX FORMULATION TIMES

TYPE	NUMBER	ENAME	TOTAL CP	AVE CP
------	--------	-------	----------	--------

1	1	PLANE182	0.000	0.000000
2	1	PLANE182	0.000	0.000000
3	1	PLANE182	0.000	0.000000
4	1	PLANE182	0.000	0.000000
5	2	SURF153	0.000	0.000000
6	1	SURF153	0.000	0.000000
7	2	CLOAD201	0.000	0.000000

Time at end of element matrix formulation CP = 0.5625.

ALL CURRENT ANSYS DATA WRITTEN TO FILE NAME= file.rdb  
 FOR POSSIBLE RESUME FROM THIS POINT

```

SPARSE MATRIX DIRECT SOLVER.
Number of equations =      12,      Maximum wavefront =      12
Memory allocated for solver      =      0.056 MB
Memory required for in-core solution      =      0.053 MB
Memory required for out-of-core solution =      0.053 MB

*** NOTE ***                      CP =      0.562      TIME= 19:50:38
The Sparse Matrix Solver is currently running in the in-core memory
mode. This memory mode uses the most amount of memory in order to
avoid using the hard drive as much as possible, which most often
results in the fastest solution time. This mode is recommended if
enough physical memory is present to accommodate all of the solver
data.
Sparse solver maximum pivot= 2.585639931E+11 at node 7 UX.
Sparse solver minimum pivot= 6.302891501E+10 at node 3 UX.
Sparse solver minimum pivot in absolute value= 6.302891501E+10 at node
3 UX.

*** ELEMENT RESULT CALCULATION TIMES
      TYPE      NUMBER      ENAME      TOTAL CP      AVE CP
      1          1      PLANE182      0.000      0.000000
      2          1      PLANE182      0.000      0.000000
      3          1      PLANE182      0.000      0.000000
      4          1      PLANE182      0.000      0.000000
      5          2      SURF153      0.000      0.000000
      6          1      SURF153      0.000      0.000000
      7          2      CLOAD201      0.000      0.000000

*** NODAL LOAD CALCULATION TIMES
      TYPE      NUMBER      ENAME      TOTAL CP      AVE CP
      1          1      PLANE182      0.000      0.000000
      2          1      PLANE182      0.000      0.000000
      3          1      PLANE182      0.000      0.000000
      4          1      PLANE182      0.000      0.000000
      5          2      SURF153      0.000      0.000000
      6          1      SURF153      0.000      0.000000
      7          2      CLOAD201      0.000      0.000000

*** LOAD STEP      1      SUBSTEP      1      COMPLETED.      CUM ITER =      1
*** TIME =      1.00000      TIME INC =      1.00000      NEW TRIANG MATRIX

*** ANSYS BINARY FILE STATISTICS
BUFFER SIZE USED= 16384
0.062 MB WRITTEN ON ELEMENT MATRIX FILE: file.emat
0.062 MB WRITTEN ON ELEMENT SAVED DATA FILE: file.esav
0.062 MB WRITTEN ON ASSEMBLED MATRIX FILE: file.full
0.312 MB WRITTEN ON RESULTS FILE: file.rst
***** Write FE CONNECTORS *****

WRITE OUT CONSTRAINT EQUATIONS TO FILE=
file.ce

*****
***** FINISHED SOLVE FOR LS 1 *****

*GET _WALLASOL FROM ACTI ITEM=TIME WALL VALUE= 19.8438889

PRINTOUT RESUMED BY /GOP

FINISH SOLUTION PROCESSING

***** ROUTINE COMPLETED ***** CP =      0.594

*** ANSYS - ENGINEERING ANALYSIS SYSTEM      RELEASE 2020 R2      20.2      ***
ANSYS Mechanical Enterprise
20120530 VERSION=WINDOWS x64      19:50:38      MAY 24, 2025 CP=      0.609

WB_femKetCauPhangBtl--Static Structural (D4)

***** ANSYS RESULTS INTERPRETATION (POST1) *****

*** NOTE ***                      CP =      0.609      TIME= 19:50:38
Reading results into the database (SET command) will update the current
displacement and force boundary conditions in the database with the
values from the results file for that load set. Note that any
subsequent solutions will use these values unless action is taken to
either SAVE the current values or not overwrite them (/EXIT,NOSAVE).

Import Data From File file.full (Format=FULL)

Import The STIFF Matrix (12 by 12)

Import Data From File file.full (Format=FULL)

Import The STIFF Matrix (12 by 12)

Import Data From File file.full (Format=FULL)

Import The RHS Matrix (12 by 1)

*DO LOOP ON PARAMETER= I FROM 1.0000      TO 6.0000      BY 1.0000

Import Data From File file.emat (Format=EMAT)

*ENDDO INDEX= I

Import Data From File file.full (Format=FULL)

*** ERROR ***                      CP =      0.703      TIME= 19:50:38
DMA Command: This matrix (MASS) is not available in the file
file.full.

```

NUMBER OF WARNING MESSAGES ENCOUNTERED= 2  
 NUMBER OF ERROR MESSAGES ENCOUNTERED= 1

+----- A N S Y S S T A T I S T I C S -----+

Release: 2020 R2 Build: 20.2 Update: UP20200601 Platform: WINDOWS x64  
 Date Run: 05/24/2025 Time: 19:50 Process ID: 15048  
 Operating System: Windows 10 (Build: 19045)

Processor Model: Intel(R) Xeon(R) CPU E3-1505M v5 @ 2.80GHz

Compiler: Intel(R) FORTRAN Compiler Version 19.0.0 (Build: 20190206)  
 Intel(R) C/C++ Compiler Version 19.0.0 (Build: 20190206)  
 Intel(R) Math Kernel Library Version 2020.0.0 Product Build 20191125

Number of machines requested : 1  
 Total number of cores available : 8  
 Number of physical cores available : 4  
 Number of processes requested : 1  
 Number of threads per process requested : 2  
 Total number of cores requested : 2 (Shared Memory Parallel)

GPU Acceleration: Not Requested

Job Name: file  
 Input File: dummy.dat  
 Working Directory: D:\LOCAL\_prgWS\AdvanceNumericalMethod\\_ProjectScratch\Scr5937

Total CPU time for main thread : 0.8 seconds  
 Total CPU time summed for all threads : 0.7 seconds

Elapsed time spent obtaining a license : 0.9 seconds  
 Elapsed time spent pre-processing model (/PREP7) : 0.0 seconds  
 Elapsed time spent solution - preprocessing : 0.0 seconds  
 Elapsed time spent computing solution : 0.1 seconds  
 Elapsed time spent solution - postprocessing : 0.0 seconds  
 Elapsed time spent post-processing model (/POST1) : 0.0 seconds

Equation solver used : Sparse (symmetric)  
 Equation solver computational rate : 0.0 Gflops

Maximum total memory used : 10.0 MB  
 Maximum total memory allocated : 2112.0 MB  
 Total physical memory available : 48 GB

Total amount of I/O written to disk : 0.0 GB  
 Total amount of I/O read from disk : 0.0 GB

+----- E N D A N S Y S S T A T I S T I C S -----+

```

*-----*
|
|               ANSYS RUN COMPLETED
|
|-----|
| Ansys 2020 R2      Build 20.2      UP20200601      WINDOWS x64 |
|-----|
| Database Requested(-db) 1024 MB  Scratch Memory Requested 1024 MB |
| Maximum Database Used 1 MB      Maximum Scratch Memory Used 9 MB |
|-----|
|
| CP Time (sec) = 0.734      Time = 19:50:38
| Elapsed Time (sec) = 2.000      Date = 05/24/2025
|
*-----*

```