Ansys 2021 R2
Engineering What's Ahead.

©2021 ANSYS, Inc.
All Rights Reserved.
Unauthorized use, distribution or duplication is prohibited.

Programmer's Reference

Ansys
ANSYS, Inc.
Southpointe
2600 Ansys Drive
Canonsburg, PA 15317
ansysinfo@ansys.com
http://www.ansys.com
(T) 724-746-3304
(F) 724-514-9494

Release 2021 R2
July 2021
002328

ANSYS, Inc. and Ansys Europe, Ltd. are UL registered ISO 9001: 2015 companies.

Copyright and Trademark Information

© 2021 ANSYS, Inc. Unauthorized use, distribution or duplication is prohibited.

Ansys, Ansys Workbench, AUTODYN, CFX, FLUENT and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries located in the United States or other countries. ICEM CFD is a trademark used by ANSYS, Inc. under license. CFX is a trademark of Sony Corporation in Japan. All other brand, product, service and feature names or trademarks are the property of their respective owners. FLEXlm and FLEXnet are trademarks of Flexera Software LLC.

Disclaimer Notice

THIS ANSYS SOFTWARE PRODUCT AND PROGRAM DOCUMENTATION INCLUDE TRADE SECRETS AND ARE CONFIDENTIAL AND PROPRIETARY PRODUCTS OF ANSYS, INC., ITS SUBSIDIARIES, OR LICENSORS. The software products and documentation are furnished by ANSYS, Inc., its subsidiaries, or affiliates under a software license agreement that contains provisions concerning non-disclosure, copying, length and nature of use, compliance with exporting laws, warranties, disclaimers, limitations of liability, and remedies, and other provisions. The software products and documentation may be used, disclosed, transferred, or copied only in accordance with the terms and conditions of that software license agreement.

ANSYS, Inc. and Ansys Europe, Ltd. are UL registered ISO 9001: 2015 companies.

U.S. Government Rights

For U.S. Government users, except as specifically granted by the ANSYS, Inc. software license agreement, the use, duplication, or disclosure by the United States Government is subject to restrictions stated in the ANSYS, Inc. software license agreement and FAR 12.212 (for non-DOD licenses).

Third-Party Software

See the legal information in the product help files for the complete Legal Notice for Ansys proprietary software and third-party software. If you are unable to access the Legal Notice, contact ANSYS, Inc.

Published in the U.S.A.

Table of Contents

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

List of Figures

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

List of Tables

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Preface

About the *Programmer's Reference* (p. 1)

The *Programmer's Reference* (p. 1) provides information about the various programming interfaces available to customers. This manual assumes that you have at least a basic knowledge of programming (a working knowledge of FORTRAN would be very helpful). The two part manual includes:

Part I - Guide to Interfacing with ANSYS

This guide describes a group of utilities as well as a set of FORTRAN routines that you can use to directly access the ANSYS database. You can also use these capabilities to access data in any of the binary files that Mechanical APDL writes or uses.

Part II - Guide to User-Programmable Features

Mechanical APDL provides a set of FORTRAN functions and routines that are available to extend or modify the program's capabilities. Using these routines requires relinking the Mechanical APDL program, resulting in a custom version of Mechanical APDL. The program provides an external commands capability which you can use to create shared libraries available to Mechanical APDL (either from ANSI standard C or FORTRAN). You can use this feature to add custom extensions to Mechanical APDL without the need to rebuild the Mechanical APDL executable.

In addition, you can find the *Ansys Parametric Design Language Guide* as part of the Mechanical APDL Help system. This guide was designed for Mechanical APDL users who have some programming skills and wish to leverage the power of the ANSYS Parametric Design Language (APDL) to increase the productivity. APDL is a scripting language that is very similar to FORTRAN. The guide describes how to define parameters (variables), how to create macro programs using APDL, how to use APDL for simple user interaction, how to encrypt an APDL macro, and how to debug an APDL macro.

Note:

The *Programmer's Reference* (p. 1) is offered solely as an aid, and does not undergo the same rigorous verification as the Mechanical APDL product documentation set. Therefore, the *Programmer's Reference* (p. 1) is not considered to be part of the formal program specification as stated in your license agreement.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Part 1: Guide to Interfacing with Ansys

Chapter 1: Format of Binary Data Files

Mechanical APDL writes several binary files to store data during an analysis. These files are named `Jobname.ext`, where `Jobname` is the name of the analysis that caused the file to be generated and `.ext` is an extension indicating the type of data in the file.

The following Binary Data File topics are available in this chapter:

1.1. Understanding Mechanical APDL Binary Files

Mechanical APDL-written binary files include the following:

- The following results files, in which Mechanical APDL stores the results of solving finite element analysis problems:
  - *Jobname*.RST A structural or coupled-field analysis
  - *Jobname*.RTH A thermal analysis
  - *Jobname*.RMG A magnetic analysis
- The *Jobname*.MODE file, storing data related to a modal analysis
- The *Jobname*.RDSP file, storing data related to a mode-superposition transient analysis.
- The *Jobname*.RFRQ file, storing data related to a mode-superposition harmonic analysis
- The *Jobname*.EMAT file, storing data related to element matrices
- The *Jobname*.SUB file, storing data related to substructure matrices
- The *Jobname*.FULL file, storing the full stiffness-mass matrix

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The *Jobname*.DSUB file, storing displacements related to substructure matrices

The files listed above cover almost all users' needs, although there are others. For more information, see the *Basic Analysis Guide*.

1.1.1. Conventions Used to Describe Binary Files

In the information describing the binary file formats:

- Record ID is the identifier for this record. Not all records will have identifiers; they're indicated only for records whose record pointers are stored in a header.
- Type indicates what kind of information this record stores.
- Number of records indicates how many records of this description are found here.
- Record length indicates the number of items stored in the record.

In some record descriptions, actual variable names used may appear in the record contents area.

1.1.2. The Standard Header for Mechanical APDL Binary Files

Each of the Mechanical APDL program's binary files contains a standard, 100-integer file header that describes the file contents. The header contains the items listed below, always in the order shown:

Item 1	The file number
Item 2	The file format. This item has a value of 1 if the file is small format, -1 if large format.
Item 3	The time, in compact form (that is, 130619 is 13:06:19)
Item 4	The date, in compact form (that is, 20041023 is 10/23/2004)
Item 5	The units of measurement used. The value of this item is as follows: • 0 for user-defined units • 1 for SI units • 2 for CSG units • 3 for U. S. Customary units (feet) • 4 for U. S. Customary units (inches) • 5 for MKS units • 6 for MPA units • 7 for μMKS units
Item 10	The Mechanical APDL release level in integer form ("*X.X*" in character form)
Item 11	The date of the Mechanical APDL release
Items 12-14	The machine identifier in integer form (three four-character strings)

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Items 15-16	The *Jobname* in integer form (two four-character strings)
Items 17-18	The Mechanical APDL product name in integer form (two four-character strings)
Item 19	The Mechanical APDL special-version label in integer form (one four-character string)
Items 20-22	The user name in integer form (three four-character strings)
Items 23-25	The machine identifier in integer form (three four-character strings)
Item 26	The system record size
Item 27	The maximum file length
Item 28	The maximum record number
Item 29	The number of cores used with shared-memory parallel
Item 30	The number of cores used with distributed-memory parallel
Items 31-38	The *Jobname* (eight four-character strings)
Items 41-60	The main analysis title in integer form (20 four-character strings)
Items 61-80	The first subtitle in integer form (20 four-character strings)
Item 81	File compression level
Item 82	File sparsification key
Item 95	The split point of the file (0 means the file will not split)
Items 97-98	LONGINT of the maximum file length

1.2. Description of the Results File

The next few pages describe the format of the Mechanical APDL results file. (In the following tables, records with a record ID containing an asterisk (*) are those you can read and store into the ANSYS database via the **LDREAD** command.)

Note: The pointers in the solution data headers are relative, not absolute pointers. For example, the 12th item in the solution data header will be relative to a position in the Data Set Index (ptrESL = DSI(*i*) + ptrESL).

This section explains the contents of the results file; that is, those files with the following extensions:

`.rmg`
`.rst`
`.rth`
`.lnn`

The ***XPL** command enables you to explore the contents of certain ANSYS binary files, including the results file. For more information, see Appendix B: Using APDL to List File Structure and Content in the *Ansys Parametric Design Language Guide*.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1.2.1. Nomenclature

A load case contains the results for an instance in an analysis. A load case is defined by a load step number and a substep number. A load case is also categorized by a cumulative iteration number and time (or frequency) values. A load case is identified by all three methods in the results file.

The results file does not have to contain all the load cases of an analysis.

A data set is used in this chapter to designate a load case.

For a complex analysis, there will be two data sets for each load case. The first data set contain the real solution and the second contains the imaginary solution.

1.2.2. Standard ANSYS File Header

See The Standard Header for Mechanical APDL Binary Files (p. 4) for a description of this set. File number (Item 1) is 12.

1.2.3. Results File Format

```
*comdeck,fdresu

c *** Copyright ANSYS.  All Rights Reserved.
c *** ansys, inc

c     **********  description of results file  **********
c     --- used for the following files:
c       .rmg
c       .rst
c       .rth
c       .lnn(lxx)

      character*8  RSTNM
      parameter   (RSTNM='rst     ')

      LONGINT         resufpL, adrZipL, resuRfpL
      integer         resubk, resuut, resuRbk, resuRut
      common /fdresu/ resufpL, adrZipL, resubk, resuut,
     x                 resuRfpL, resuRbk, resuRut

c   ********** common variable descriptions ***********
co  resufpL      file position on file resu
co  resubk       block number for file resu  (usually 6)
co  resuut       file unit for file resu     (0 if not open)  FUN12
c0  resuRxx      variables for remote modal RST file
c   See fddesc for documentation of how binary files are stored

c     **********  file format  **********

c        recid tells the identifier for this record.  Not all records will have
c             identifiers -- they are only indicated for those records whose
c             record pointers are stored in a header.

c        type tells what kind of information is stored in this record:
c             i - integer
c             dp - double precision
c             cmp - complex

c        nrec tells how many records of this description are found here

c        lrec tells how long the records are (how many items are stored)
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c recid    type    nrec    lrec     contents

c  ---      i       1      100      standard ANSYS file header (see binhed8 for
c                                   details of header contents)

c  ---      i       1       80      .RST FILE HEADER

c                                    fun12,    maxn,     nnod,   resmax,   numdof,
c                                     maxe,    nelm,      kan,    nsets,   ptrend,  (10)
c                                  ptrDSIl, ptrTIMl,  ptrLSPl,  ptrELMl,  ptrNODl,
c                                   ptrGEO, ptrCYCl,   CMSflg,    csEls,    units,  (20)
c                                  nSector,  csCord,  ptrEnd8,  ptrEnd8,  fsiflag,
c                                        0,  noffst,   eoffst,   nTrans, ptrTRANl,  (30)
c                                 PrecKey,   csNds,   cpxrst,   extopt,   nlgeom,
c                                AvailDatal,  mmass, kPerturb,  XfemKey,  rstsprs,  (40)
c                                  ptrDSIh, ptrTIMh,  ptrLSPh,  ptrCYCh,  ptrELMh,
c                                  ptrNODh, ptrGEOh, ptrTRANh,  Glbnnod, ptrGNODl,  (50)
c                                 ptrGNODh, qrDmpKy,  MSUPkey,   PSDkey,cycMSUPkey,
c                           XfemCrkPropTech,cycNoDup,decompMth, nProcSol,  mpiWrld,  (60)
c                                  AMtype,udfrqkey,ptrUDFRQl,ptrUDFRQh,   cpxeng,
c                               AvailDatah,       0,        0,        0,        0,  (70)
c                                        0,       0,        0,        0,        0,
c                                        0,       0,        0,        0,        0   (80)

c                                  each item in header is described below:

c                                   fun12  - unit number (resu file is 12)
c                                   maxn   - maximum node number of the model
c                                   nnod   - the actual number of nodes used in
c                                            the solution phase
c                                   resmax - the maximum number of data sets
c                                            allowed on the file (defaults to
c                                            10000; minimum allowed is 10)
c                                   numdof - number of DOFs per node
c                                   maxe   - maximum element number of the
c                                            finite element model
c                                   nelm   - number of finite elements
c                                   kan    - analysis type
c                                   nsets  - number of data sets on the file
c                                   ptrend - pointer to the end of the file
c                                             (see ptrEnd8 in 23,24)
c                                ptrDSIl,h - 64 bit pointer to the data steps
c                                            index table
c                                ptrTIMl,h - 64 bit pointer to the table of time
c                                            values for a load step
c                                ptrLSPl,h - 64 bit pointer to the table of load
c                                            step, substep, and cumulative
c                                            iteration numbers
c                                ptrELMl,h - 64 bit pointer to the element equivalence
c                                            table (used when the mesh does not
c                                            change during solution)
c                                ptrNODl,h - 64 bit pointer to the nodal equivalence
c                                            table (used when the mesh does not
c                                            change during solution)
c                                ptrGEOl,h - 64 bit pointer to the beginning of
c                                            geometry information (used when the
c                                            mesh does not change during solution)
c                                ptrCYCl,h - 64 bit pointer to the table of cyc sym
c                                            nodal-diameters at each load step
c                                   CMSflg - CMS results flag: 0-non cms, >0-cms
c                                   csEls  - Cyclic sym # eles in master sector
c                                   units  - unit system used
c                                            =-1 - no /UNITS specification
c                                            = 0 - user defined units
c                                            = 1 - SI
c                                            = 2 - CSG
c                                            = 3 - U.S. Customary, using feet
c                                            = 4 - U.S. Customary, using inches
c                                            = 5 - MKS
c                                            = 6 - MPA
c                                            = 7 - uMKS
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                  nSector - number of sectors for cyclic sym
c                                  csCord  - Cyclic symmetry coordinate system
c                                  ptrEnd8 - 64 bit file pointer to the end of
c                                            the file (i.e., length of file)
c                                  fsiflag - FSI analyis flag
c                                  noffst  - node offset used in writing file
c                                  eoffst  - elem offset used in writing file
c                                  nTrans  - number of SE transformation vects
c                                ptrTRANl,h - 64 bit pointer to SE transformation vects
c                                  PrecKey - 0, double precision
c                                            1, single for element results only
c                                            2, single for all data
c                                  csNds   - Cyclic sym # nds in master sector
c                                  cpxrst  - complex results flag (0-no, 1-yes)
c                                  extopt  - mode extraction option
c                                  nlgeom  - NLGEOM key
c                             AvailDatal,h  - bits indicating available data any
c                                            where on the file; see resucm.inc
c                                  mmass   - number of missing mass resp. present
c                                 kPerturb - key for Linear Perturbation results
c                                  XfemKey - XFEM flag (set equal to Active_XfenId)
c                                                     (0=Inactive, 1=Active)
c                                  rstsprs - bitmask for suppressed items
c                                  Glbnnod - global number of nodes actually used
c                                            in the solution phase (== nnod unless
c                                            using Distributed Ansys)
c                                ptrGNODl,h - 64 bit pointer to the global nodal
c                                            equivalence table (only used with
c                                            Distributed ANSYS and when the mesh
c                                            does not change during solution)
c                                  qrDmpKy - QR damped calculations key
c                                  MSUPkey - MSUP results expanded with MSUPCombineModes
c                                  PSDkey  - PSD key for element results combination
c                                            = 1 flag PSD analysis
c                                            = 2 flag single/multi-point response spectrum
c                                cycMSUPkey - rst file format is for subsequent cyclic MSUP
c                                            (only base results on file)
c                            XfemCrkPropTech - XFEM crack propagation technology
c                                                (0=phantom node-based, 1=singularity-based)
c                                  cycNoDup - no duplicate sector created
c                                 decompMth - domain decomposition method employed with DMP
c                                            (>= 0 = MESH; -1 = FREQ, -2 = CYCHI)
c                                  nProcSol - when the domain decomposition method for DMP
c                                            is FREQ or CYCHI --> number of processes used for
c                                            each frequency or cyclic harmonic index solution
c                                   mpiWrld - unique identifier for determining which results
c                                            file belongs to
c                                    AMtype - additive manfacturing (AM) flag
c                                  udfrqkey - Key for format and writing of undamped frequency record
c                         ptrUDFRQl,ptrUDFRQh - pointer to undamped frequency record
c                                    cpxeng - 0 = default; lrec = 11
c                                            1 = element ENG record contains additional quantities;
c                                                lrec = 21
c
c                                    Note: ptrXXX are relative to beginning of file

c   ---      i       1     numdof   Degrees of freedom per node
c                                    DOF reference numbers are:
c         UX  = 1, UY  = 2, UZ  = 3, ROTX= 4, ROTY= 5, ROTZ= 6, AX  = 7, AY  = 8
c         AZ  = 9, VX  =10, VY  =11, VZ  =12, GFV1=13, GFV2=14, GFV3=15, WARP=16
c         CONC=17, HDSP=18, PRES=19, TEMP=20, VOLT=21, MAG =22, ENKE=23, ENDS=24
c         EMF =25, CURR=26, SP01=27, SP02=28, SP03=29, SP04=30, SP05=31, SP06=32
c         TBOT=33, TE2 =34, TE3 =35, TE4 =36, TE5 =37, TE6 =38, TE7 =39, TE8 =40
c         TE9 =41, TE10=42, TE11=43, TE12=44, TE13=45, TE14=46, TE15=47, TE16=48
c         TE17=49, TE18=50, TE19=51, TE20=52, TE21=53, TE22=54, TE23=55, TE24=56
c         TE25=57, TE26=58, TE27=59, TE28=60, TE29=61, TE30=62, TE31=63, TTOP=64
c                                     (curdof(i),i=1,numdof)

c   NOD      i       1     nnod     Nodal equivalence table. This table equates
c                                    the number used for storage to the actual
c                                    node number
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                    (Back(i),i=1,nnod)

c   ELM      i       1     nelm     Element equivalence table. The ANSYS program
c                                   stores all element data in the numerical
c                                   order that the SOLUTION processor solves the
c                                   elements.  This table equates the order
c                                   number used to the actual element number

c   GNOD     i       1    Glbnnod   Global nodal equivalence table. This table
c                                   equates the number used for storage to the
c                                   actual node number. Only written by the
c                                   master process in Distributed Ansys
c                                    (GlbBack(i),i=1,Glbnnod)

c   DSI      i       1    2*resmax  Data sets index table. This record contains
c                                   the record pointers for the beginning of
c                                   each data set. The first resmax records are
c                                   the first 32 bits of the index, the second
c                                   resmax records are the second 32 bits. To
c                                   create the 64 bit pointer, use:
c                                     LONGPTR = largeIntGet (first,second)
c                                   Read the solution data header as follows:
c                                     call bioBasePut (nblk,LONGPTR)
c                                     loc = bioiqr (nblk,12)
c                                     call biord (nblk,loc,...
c                                   The rest of the file reading continues to use
c                                   the ptrXXX's that are in the headers.

c   TIM     dp       1     resmax   Time/freq table. This record contains the
c                                   time (or frequency) values for each data
c                                   set.

c   UDFRQ   dp       1     resmax   Undamped freq table. This record contains the
c                                   undamped frequency values for each data
c                                   set in a QRDAMPED modal analysis. This record
c                                   exists only when qrDmpKy = 1
c                                   (see the results file header for qrDmpKy)

c   LSP      i       1    3*resmax  Data set identifiers. This record contains
c                                   the load step, substep, and cumulative
c                                   iteration numbers for each data set.

c   CYC      i       1     resmax   Cyclic symmetry harmonic index

c   TRAN     dp    nTran     25     Substructure transformation vectors

c   GEO      i       1       80     Geometry data header

c                                        0,  maxety,   maxrl,    nnod,    nelm,
c                                   maxcsy,  ptrETY,  ptrREL,  ptrLOC,  ptrCSY,  (10)
c                                   ptrEID,  maxsec,  secsiz,  nummat,  matsiz,
c                                   ptrMAS,  csysiz,  elmsiz,  etysiz,   rlsiz,  (20)
c                                  ptrETYl, ptrETYh, ptrRELl, ptrRELh, ptrCSYl,
c                                  ptrCSYh, ptrLOCl, ptrLOCh, ptrEIDl, ptrEIDh,  (30)
c                                  ptrMASl, ptrMASh, ptrSECl, ptrSECh, ptrMATl,
c                                  ptrMATh, ptrCNTl, ptrCNTh, ptrNODl, ptrNODh,  (40)
c                                  ptrELMl, ptrELMh, Glblenb,ptrGNODl,ptrGNODh,
c                                     maxn,NodesUpd,  lenbac, numcomp, mxcmpsz,  (50)
c                                 ptrCOMPl,ptrCOMPh,nMatProp,  nStage, maxMSsz,
c                                   ptrMSl,  ptrMSh,   nCycP,ptrCycPl,ptrCycPh,  (60)
c                                   numety,   numrl,  numcsy,  numsec, mapFlag,
c                                  cysCSID,       0,       0,       0,       0,  (70)
c                                        0,       0,       0,       0,       0,
c                                        0,       0,       0,       0,       0   (80)

c                                  each item in header is described below:

c                                   0      - position not used
c                                   maxety - the maximum element type reference
c                                            number in the model
c                                   maxrl  - the maximum real constant reference
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                          number in the model
c                                nnod   - the number of defined nodes in the
c                                          model
c                                nelm   - the number of defined elements in
c                                          the model
c                                maxcsy - the maximum coordinate system
c                                          reference number in the model
c                                ptrETY - pointer to the element type index
c                                          table
c                                ptrREL - pointer to the real constant
c                                          index table
c                                ptrLOC - pointer to the nodal point
c                                          locations
c                                ptrCSY - pointer to the local coordinate
c                                          system index table
c                                ptrEID - pointer to the element index
c                                          table
c                                maxsec - the maximum section
c                                          reference number in the model
c                                secsiz - the maximum size that any
c                                          section record may have
c                                nummat - the number of materials
c                                          in the model
c                                matsiz - the maximum size that any material
c                                          property or table may have
c                                ptrMAS - pointer to the diagonal mass matrix
c                                csysiz - the number of items describing a
c                                          local coordinate system (usually
c                                          24)
c                                elmsiz - the maximum number of nodes that a
c                                          defined element may have
c                                etysiz - the number of items describing an
c                                          element type(=IELCSZ from echprm.inc)
c                                rlsiz  - the maximum number of items
c                                          defining a real constant (0, if no
c                                          real constants are defined)
c                             ptrETYl,h - 64 bit pointer to element type data
c                             ptrRELl,h - 64 bit pointer to real constant data
c                             ptrCSYl,h - 64 bit pointer to coordinate system data
c                             ptrLOCl,h - 64 bit pointer to nodal locations
c                             ptrEIDl,h - 64 bit pointer to element data
c                             ptrSECl,h - 64 bit pointer to section data
c                             ptrMATl,h - 64 bit pointer to material data
c                             ptrCNTl,h - 64 bit pointer to element centroids
c                             ptrNODl,h - 64 bit pointer to nodal equivalence table
c                             ptrELMl,h - 64 bit pointer to element equivalence table
c                               Glblenb - global number of nodes actually used
c                                          in the solution phase (== lenbac unless
c                                          using Distributed Ansys)
c                             ptrGNODl,h- 64 bit pointer to the global nodal
c                                          equivalence table (only used with
c                                          Distributed ANSYS and when the mesh
c                                          does not change during solution)
c                                maxn   - maximum node number of the model
c                              NodesUpd - 1, node coords have been updated
c                                lenbac - the actual number of nodes used in
c                                          the solution phase
c                                numcomp - number of components/assemblies stored
c                                          (only node/elem components/assemblies)
c                                mxcmpsz - maximum size (in integer words) that any
c                                          component/assembly record may have
c                             ptrCOMPl,h - 64 bit pointer to component/assembly data
c                               nMatProp - number of properties stored per material
c                                  nStage - number of stages
c                                maxMSsz - maximum size (in integer words) that a
c                                          stage record can have
c                                ptrMSl,h - 64 bit pointer to multistage (MS) cyclic
c                                          analysis data
c                                   nCycP - number of cyclic edge node pair tables (CYCLIC and MS)
c                         ptrCycPl, ptrCycPh - pointers to cyclic edge node pair tables
c                                numety - the number of defined element types
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                          in the model
c                                   numrl  - the number of defined real
c                                          constants in the model
c                                   numcsy - the number of defined coordinate
c                                          systems in the model
c                                   numsec - the number of defined sections in
c                                          the model
c                                  mapFlag - flag to indicate format of mapping
c                                          index vectors for element types,
c                                          real constants, coordinate systems,
c                                          and sections.
c                                          = 0, old format with 1 vector of
c                                               maxDef len
c                                          = 1, new format with 2 vectors
c                                               each having numDef len
c                                  cycCSID - coordinate system number (CYCLIC and MS)
c                                          = 0 is ignored (must be cylindrical)
c
c                                   Note: ptrXXX are relative to beginning of file

c   ETY      i       1     varies   if mapFlag == 0 --> Length = maxety
c                                   The element types index table. This record
c                                   contains record pointers for each element
c                                   type description.
c                                  (Relative to ptrETYPL)
c
c                                   if mapFlag == 1 --> Length = numety
c                                   Elemnt type external number mapping.
c                                   This record maps the number used for storage
c                                   to the actual element type number

c  ---       i       1     numety   Only exists if mapFlag ==1
c                                   The element types index table. This record
c                                   contains record pointers for each element
c                                   type description.
c                                  (Relative to ptrETYPL)

c   ---      i     numety  etysiz   Element type description. Each of these
c                                   records is pointed to by a record pointer
c                                   given in the record labeled ETY. See
c                                   routines echprm and elccmt for a complete
c                                   description of the items stored here.

c                                   These items are typically stored into the
c                                   IELC array, and are used to determine the
c                                   element type characteristics at runtime.
c                                   The following items are typically of
c                                   interest:
c                                   * Item 1     - element type reference number
c                                   * Item 2     - element routine number
c                                   * Items 3-14 - element type option keys
c                                                  (keyopts)
c                                   * Item 34    - DOF/node for this element
c                                                  type.  This is a bit mapping
c                                                  of the DOF/node.
c                                   * Item 61    - number of nodes for this
c                                                  element type (nodelm)
c                                   * Item 63    - number of nodes per element
c                                                  having nodal forces, etc.
c                                                  (nodfor)
c                                   * Item 94    - number of nodes per element
c                                                  having nodal stresses, etc.
c                                                  (nodstr).  This number is the
c                                                  number of corner nodes for
c                                                  higher-ordered elements.
c                                                  NOTE- the /config NST1 option
c                                                  may suppress all but one node
c                                                  output of nodal ENS EEL EPL
c                                                  ECR ETH and ENL for any given
c                                                  element.
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c   REL      i       1     varies   if mapFlag == 0 --> Length = maxrl
c                                   Real constants index table.  The record
c                                   contains record pointers for each real
c                                   constant set.
c                                  (Relative to ptrRELL)
c
c                                   if mapFlag == 1 --> Length = numrl
c                                   Real constants external number mapping.
c                                   This record maps the number used for storage
c                                   to the actual real constant number

c  ---       i       1      numrl   Only exists if mapFlag ==1
c                                   Real constants index table.  The record
c                                   contains record pointers for each real
c                                   constant set.
c                                  (Relative to ptrRELL)

c   ---     dp     numrl   varies   Element real constant data.  These records
c                                   contain real constant data used for the
c                                   elements.  (See the ANSYS Elements Reference
c                                   manual for values for a specific element.)
c                                   Each of these records is pointed to by a
c                                   record pointer given in the record labeled
c                                   REL. The length of these records varies for
c                                   each element type (actual length is returned
c                                   from routine BINRD8).

c   CSY      i       1     varies   if mapFlag == 0 --> Length = maxcsy
c                                   Coordinate systems index table.  This record
c                                   contains the record pointers for each
c                                   coordinate system set. The ANSYS program
c                                   writes coordinate systems only if local
c                                   coordinate systems were defined.  If a local
c                                   system was defined, the predefined global
c                                   systems 1 to 2 also will be written.  The
c                                   global Cartesian system 0 will never be
c                                   written.
c                                  (Relative to ptrCSYSL)
c
c                                   if mapFlag == 1 --> Length = numcsy
c                                   Coordinate system external number mapping.
c                                   This record maps the number used for storage
c                                   to the actual coordinate system number

c  ---       i       1     numcsy   Only exists if mapFlag ==1
c                                   Coordinate systems index table.  This record
c                                   contains the record pointers for each
c                                   coordinate system set. The ANSYS program
c                                   writes coordinate systems only if local
c                                   coordinate systems were defined.  If a local
c                                   system was defined, the predefined global
c                                   systems 1 to 2 also will be written.  The
c                                   global Cartesian system 0 will never be
c                                   written.
c                                  (Relative to ptrCSYSL)

c   ---     dp     numcsy  csysiz   Coordinate system description.  These
c                                   records contain coordinate system data for
c                                   each coordinate system defined.  Each of
c                                   these records is pointed to by a record
c                                   pointer given in the record labeled SYS.

c                                   The items stored in each record:

c                                   * Items 1-9  are the transformation matrix.
c                                   * Items 10-12 are the coordinate system
c                                     origin (XC,YC,ZC).
c                                   * Items 13-14 are the coordinate system
c                                     parameters (PAR1, PAR2).
c                                   * Items 16-18 are the angles used to define
c                                     the coordinate system.
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                    * Items 19-20 are theta and phi singularity
c                                      keys.
c                                    * Item 21 is the coordinate system type
c                                      (0, 1, 2, or 3).
c                                    * Item 22 is the coordinate system reference
c                                      number.

c   LOC     dp     nnod      7       Node,X,Y,Z,THXY,THYZ,THZX for each node
c                                    Nodes are in node number order

c   EID      i       1    2*nelm     Element descriptions index table. This
c                                    record contains the record pointers for each
c                                    element description.  The first nelm values
c                                    are the lower 32 bits and the second nelm
c                                    values are the higher 32 bits. The order
c                                    of the elements is the same as the order
c                                    in the element equivalence table.
c                                   (Relative to ptrEIDL)

c   ---      i     nelm  10+nodelm  Element descriptions. Each of these records
c                                    is pointed to by a record pointer given in
c                                    the record labeled EID. The length of these
c                                    records varies for each element (actual
c                                    length is returned from routine BINRD8).
c                                    nodelm shown here is the number of nodes for
c                                    this element. Its value is defined in the
c                                    element type description record.

c                                    The items stored in each record:
c                                       mat,  type,  real, secnum, esys,
c                                     death,solidm, shape, elnum,  baseeid,
c                                     nodes

c                                   each item is described below:

c                                      mat    - material reference number
c                                      type   - element type number
c                                      real   - real constant reference number
c                                      secnum - section number
c                                      esys   - element coordinate system
c                                      death  - death flag
c                                               = 0 - alive
c                                               = 1 - dead
c                                      solidm - solid model reference
c                                      shape  - coded shape key
c                                      elnum  - element number
c                                      baseeid- base element number
c                  (applicable to reinforcing elements only)
c                                      nodes  - node numbers defining the element
c                                               (See the ANSYS Elements Reference
c                                               for nodal order of an element)

c   CENT    dp     nelm      6       Centroid record for each element

c   MAS     dp       1  nnod*numdof Diagonal mass matrix

c   SEC      i       1     varies    if mapFlag == 0 --> Length = maxsec
c                                    Section index table.  The record
c                                    contains record pointers for each
c                                    section set.
c
c                                    if mapFlag == 1 --> Length = numsec
c                                    Section external number mapping.
c                                    This record maps the number used for storage
c                                    to the actual section number

c  ---       i       1     numsec    Only exists if mapFlag ==1
c                                    Section index table.  The record
c                                    contains record pointers for each
c                                    section set.
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c   ---     dp     numsec  varies   Element section data.  These records
c                                   contain section data used for the
c                                   elements.
c                                   Each of these records is pointed to by a
c                                   record pointer given in the record labeled
c                                   SEC. The length of these records varies for
c                                   each section type (actual length is returned
c                                   from routine BINRD8).

c   MAT      i     1  3+nummat*(nMatProp+1)+2
c                                   Total Sz = Header Data + (nMatProp+1)*Number of Materials + Tail
c                                   The 1st 3 integers contain the header information
c                                   1) Version number (-101 for differentiation from prev rst data)
c                                   2) Header size which is 3
c                                   3) Size of the index array
c
c                                   Ith material ID is stored at Data(3+((I-1)*(nMatProp+1))+1)
c                                     I varies from 1 to nummat
c
c                                   Material record pointers for the material ID at Ith position
c                                     is stored from location
c                                     3+(I-1)*(nMatProp+1)+2 to 3+(I-1)*(nMatProp+1)+1+nMatProp
c                                   The record includes MP pointers followed by TB pointers for a
c                                     single Material ID
c                                   Last 2 data contains the active table number and material ID

c   ---     dp     nummat  varies   Material property data.  These records
c                                   contain material property data used for the
c                                   elements.
c                                   Each of these records is pointed to by a
c                                   record pointer given in the record labeled
c                                   MAT. The length of these records varies for
c                                   each material property type (actual length
c                                   is returned from routine BINRD8).

c   NOD      i       1    lenbac    Nodal equivalence table. This table equates
c                                   the number used for storage to the actual
c                                   node number
c                                    (Back(i),i=1,lenbac)

c   ELM      i       1     nelm     Element equivalence table. The ANSYS program
c                                   stores all element data in the numerical
c                                   order that the SOLUTION processor solves the
c                                   elements.  This table equates the order
c                                   number used to the actual element number

c   GNOD     i       1    Glblenb   Global nodal equivalence table. This table
c                                   equates the number used for storage to the
c                                   actual node number. Only written by the
c                                   master process in Distributed Ansys
c                                    (GlbBack(i),i=1,Glblenb)

c   COMP     i    numCmp   varies   Component/assembly data.  The first word will
c                                   describe the type of component or assembly
c                                   (1=node component, 2=elem component, 11->15 assembly)
c                                   For components, values 2->9 are the component
c                                   name converted to integers.  The remaining
c                                   values are the list of nodes/elems in the
c                                   component.  For assemblies, the remaining
c                                   values (in groups of 8) are the component
c                                   names converted to integers.

c     MS    nStage                  Following 2 records are repeated nStage times.
c           | i       PARMSIZEW+25  Stage char/int data.
c                                      1-PARMSIZEW packed stage name
c                                      PARMSIZEW+1 SEkey, Nsector, Hindex, Numoff
c                                                  cyclic CE min/max
c                                                  downstream interstage CE min/max
c                                                  upstream interstage CE min/max
c                                                  number of nodes, min/max node numbers (base only)
c                                                  number of elements, min/max element numbers (base only)
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

```
c           | dp             10        Stage dp data.
c                                      1-6 min/max nodal coordinates of the stage
c           i     1  nStage*nStage  Table of connection between stages

c   CYCP    i       1      varies   Record exists only if the number of tables nCycP and
c                                   associated pointers are non-zero.
c                                   nCycP = 1 for CYCLIC
c                                         = number of non-superelement and non-3D stages (MS)
c                                   Maximum record size  = number of nodes defined
c                                   Record content:
c                                      1:nCycP    = number of nodes for each table (iCycP)
c                                      nCycP+1... = node pairs of each table
c                                   Effective record size = nCycP + sum(iCycP*2)

c       The solution information is stored starting at this point in the file.
c       The remaining records on the file are repeated as a group nsets times
c       (once for each data set).  Item nsets is defined in the file header.

c       Each set of data is pointed to by a record pointer given in the record
c       labeled DSI.

c   ---      i       1       200     Solution data header.

c                                        0,    nelm,    nnod,   maskl,   itime,
c                                     iter,  ncumit,     nrf,  cs_LSC,   nmast,  (10)
c                                   ptrNSL,  ptrESL,   ptrRF,  ptrMST,   ptrBC,
c                                   rxtrap,    mode,    isym,  kcmplx,  numdof,  (20)
c                                     DOFS,    DOFS,    DOFS,    DOFS,    DOFS,
c                                     DOFS,    DOFS,    DOFS,    DOFS,    DOFS,  (30)
c                                     DOFS,    DOFS,    DOFS,    DOFS,    DOFS,
c                                     DOFS,    DOFS,    DOFS,    DOFS,    DOFS,  (40)
c                                     DOFS,    DOFS,    DOFS,    DOFS,    DOFS,
c                                     DOFS,    DOFS,    DOFS,    DOFS,    DOFS,  (50)
c                                    title,   title,   title,   title,   title,
c                                    title,   title,   title,   title,   title,  (60)
c                                    title,   title,   title,   title,   title,
c                                    title,   title,   title,   title,   title,  (70)
c                                   stitle,  stitle,  stitle,  stitle,  stitle,
c                                   stitle,  stitle,  stitle,  stitle,  stitle,  (80)
c                                   stitle,  stitle,  stitle,  stitle,  stitle,
c                                   stitle,  stitle,  stitle,  stitle,  stitle,  (90)
c                                   dbmtim,  dbmdat,  dbfncl,  soltim,  soldat,
c                                   ptrOND,  ptrOEL,       0,       0,  ptrEXT,  (100)
c                                        0,       0, ptrEXTl, ptrEXTh, ptrNSLl,
c                                  ptrNSLh,  ptrRFl,  ptrRFh, ptrMSTl, ptrMSTh,  (110)
c                                   ptrBCl,  ptrBCh,       0,       0, ptrONDl,
c                                   ptrONDh, ptrOELl, ptrOELh, ptrESLl, ptrESLh,  (120)
c                                   ptrOSLl, ptrOSLh,sizeDEAD,ptrDEADl,ptrDEADh,
c                                   PrinKey, numvdof, numadof, ptrNARl, ptrNARh,  (130)
c                                   ptrVSLl, ptrVSLh, ptrASLl, ptrASLh,   nMSHI,
c                                 ptrMSHIl,ptrMSHIh,       0,numRotCmp,      0,  (140)
c                                   ptrRCMl, ptrRCMh, nNodStr,       0,ptrNDSTRl,
c                                ptrNDSTRh,AvailDatal,geomID, ptrGEOl, ptrGEOh,  (150)
c                                    k2d3d,   maskh,  AMstep, inverse,AvailDatah,
c                                        0,       0,       0,       0, ptrCINT,  (160)
c                                  lenCINT, ptrXFEM, lenXFEM,       0,       0,
c                                        0,       0,       0,       0,       0,  (170)
c                                        0,       0,       0,       0,       0,
c                                        0,       0,       0,       0,       0,  (180)
c                                        0,       0,       0,       0,       0,
c                                        0,       0,       0,       0,       0,  (190)
c                                        0,       0,       0,       0,       0,
c                                        0,       0,       0,       0,       0   (200)

c                                   each item in header is described below:

c                                    nelm   - number of elements
c                                    nnod   - number of nodes
c                                   maskl,h - bitmask for the existence of
c                                             several records.  If a bit is set
c                                             here, it indicates that the
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                          corresponding record exists on the
c                                          file.
c                                          The items in the bitmask that
c                                          correspond to each record are shown
c                                          in the record descriptions below.
c                                  itime  - loadstep
c                                  iter   - iteration number
c                                  ncumit - cumulative iteration number
c                                  nrf    - number of reaction forces
c                                  cs_LSC - cyclic symmetry count of the
c                                           load step for this SOLVE
c                                  nmast  - number of masters
c                                  ptrNSL - 32-bit pointer to nodal solution
c                                  ptrESL - 32-bit pointer to element solution
c                                  ptrRF  - 32-bit pointer to reaction forces
c                                  ptrMST - 32-bit pointer to the masters
c                                  ptrBC  - 32-bit pointer to the boundary conditions
c                                  rxtrap - key to extrapolate integration
c                                           point results to nodes
c                                           = 0 - move
c                                           = 1 - extrapolate unless active
c                                                 non-linear
c                                           = 2 - extrapolate always
c                                  mode   - mode number of harmonic loading
c                                           (for cyclic symmetry: this is cs_LSF
c                                           = first load step for this SOLVE)
c                                  isym   - symmetry for harmonic loading
c                                           (for cyclic symmetry: this is cs_LSL
c                                           = last load step for this SOLVE)
c                                  kcmplx - complex key
c                                           = 0 - real
c                                           = 1 - imaginary
c                                  numdof - number of DOFs/nodes for this data
c                                           set
c                                  DOFS   - DOF/node reference numbers (numdof
c                                           values)
c                                  title  - main title (in integer form)
c                                 stitle1 - 1st subtitle (in integer form)
c                                  dbmtim - time (in compact form) when the
c                                           database was last modified
c                                  dbmdat - date (in compact form) when the
c                                           database was last modified
c                                  dbfncl - number of times that the database
c                                           was modified
c                                  soltim - time (in compact form) when the
c                                           solution for this data set was done
c                                  soldat - date (in compact form) when the
c                                           solution for this data set was done
c                                  ptrOND - 32-bit pointer to the ordered node
c                                           list (load case files only)
c                                  ptrOEL - 32-bit pointer to the ordered element
c                                           list (load case files only)
c                                  ptrEXT - 32-bit pointer to header extension
c                                ptrEXTl,h - 64-bit pointer to header extension
c                                ptrNSLl,h - 64-bit pointer to nodal solution
c                                ptrRFl,h  - 64-bit pointer to reaction forces
c                                ptrMSTl,h - 64-bit pointer to the masters
c                                ptrBCl,h  - 64-bit pointer to the boundary conditions
c                                ptrONDl,h - 64-bit pointer to the ordered node
c                                           list (load case files only)
c                                ptrOELl,h - 64-bit pointer to the ordered element
c                                           list (load case files only)
c                                ptrESLl,h - 64-bit pointer to element solution
c                                ptrOSLl,h - 64-bit pointer to extra solution vector
c                                           (to be used later for rezoning project)
c                                sizeDEAD  - size of dead element list
c                               ptrDEADl,h - 64-bit pointer to dead element list
c                                  PrinKey - principal stress key:
c                                           0, use rstsprs (if bit 26 set, no prin)
c                                           1, principals are written for this set
c                                          -1, no principals are written for this set
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                  numvdof - number of velocity dofs
c                                  numadof - number of acceleration dofs
c                                ptrNARl,h - 64-bit pointer to nodal averaged
c                                            results solution
c                                ptrVSLl,h - 64-bit pointer to transient velocity
c                                            solution
c                                ptrASLl,h - 64-bit pointer to transient acceleration
c                                            solution
c                                nMSHI     - length of multistage harmoninc index table
c                                ptrMSHIl,h- 64-bit pointer to multistage harmonic
c                                            index table
c                                numRotCmp - number of rotating components
c                                ptrRCM,h - 64-bit pointer to RCM
c                                  nNodStr - 0, no nodal component stresses
c                                            1, one set (TOP for shells)
c                                            2, two sets (TOP,BOT for shells)
c                                            3, three sets (TOP,BOT,MID)
c                             ptrNDSTRl,h - 64 bit pointer to nodal component str
c                            AvailDatal,h - bits indicating available data
c                                            in this data set; see resucm.inc
c                                 geomID - number identifying which geometry (mesh)
c                                            is used for this set of data (when mesh
c                                            does not change during solution this should
c                                            always be equal to 1)
c                                ptrGEOl,h - 64 bit pointer to geometry data (when mesh
c                                            does not change during solution this points
c                                            to first GEO record near start of file)
c                                   k2d3d - key indicator 3d run on for 2d 3d analysis
c                                  AMstep - additive manufacturing process step
c                                             BUILD,COOL,HIP,REMOVE,USER,HEATTREAT
c                                               2     3   4    5     6      7
c                                 inverse - 1, the result data is from an inverse analysis
c                                             0, the result data is from a forward analysis
c                                             2, the result data is from a forward analysis
c                                                   after inverse analysis
c                                 ptrCINT - pointer to CINT record
c                                 lenCINT - length of CINT record (doubles)
c                                 ptrXFEM - pointer to XFEM record
c                                 lenXFEM - length of XFEM record (doubles)
c                                       0 - position not used
c
c                                 Note: ptrXXX are relative to ptrDSI, except ptrGEO
c                                       which is relative to the beginning of the file

c   ---     dp       1        100   Solution header - double precision data

c                                    timfrq,  lfacto,  lfactn,  cptime,    tref,
c                                     tunif,       0,       0,       0, dsfpinc,  (10)
c                                     accel,   accel,   accel,   omega,   omega,
c                                     omega,   omega,   omega,   omega, omegacg,  (20)
c                                   omegacg, omegacg, omegacg, omegacg, omegacg,
c                                    cgcent,  cgcent,  cgcent, fatjack, fatjack,  (30)
c                                     dval1,   dval2,   dval3,       0,       0,
c                                         0,       0,       0,       0,       0,  (40)
c                                         0,       0,       0,       0,       0,
c                                         0,       0,       0,       0,       0,  (50)
c                                    timdat,  timdat,  timdat,  timdat,  timdat,
c                                    timdat,  timdat,  timdat,  timdat,  timdat,  (60)
c                                    timdat,  timdat,  timdat,  timdat,  timdat,
c                                    timdat,  timdat,  timdat,  timdat,  timdat,  (70)
c                                    timdat,  timdat,  timdat,  timdat,  timdat,
c                                    timdat,  timdat,  timdat,  timdat,  timdat,  (80)
c                                    timdat,  timdat,  timdat,  timdat,  timdat,
c                                    timdat,  timdat,  timdat,  timdat,  timdat,  (90)
c                                    timdat,  timdat,  timdat,  timdat,  timdat,
c                                    timdat,  timdat,  timdat,  timdat,  timdat   (100)

c                                   each item is described below:

c                                     timfrq  - time value (or frequency value,
c                                               for a modal or harmonic analysis)
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                       lfacto  - the "old" load factor (used in
c                                                 ramping a load between old and new
c                                                 values)
c                                       lfactn  - the "new" load factor
c                                       cptime  - elapsed cpu time (in seconds)
c                                       tref    - the reference temperature
c                                       tunif   - the uniform temperature
c                                       dsfpinc - incident power of diffuse sound field
c                                       accel   - linear acceleration terms
c                                       omega   - angular velocity (first 3 terms) and
c                                                 angular acceleration (second 3 terms)
c                                       omegacg - angular velocity (first 3 terms) and
c                                                 angular acceleration (second 3 terms)
c                                                 these velocity/acceleration terms are
c                                                 computed about the center of gravity
c                                       cgcent  - (x,y,z) location of center of gravity
c                                       fatjack - FATJACK ocean wave data (wave height
c                                                 and period)
c                                         dval1 - FATJACK ocean wave direction
c                                         dval2 - machine rpm
c                                         dval3 - central frequency of tune band
c                                        timdat - load data (slot 53 is substep convergence key)

c   EXT      i       1        200    Header extension
c                                       positions   1-32  - current DOF for this
c                                                           result set
c                                       positions  33-64  - current DOF labels for
c                                                           this result set
c                                       positions  65-84  - The third title, in
c                                                           integer form
c                                       positions  85-104 - The fourth title, in
c                                                           integer form
c                                       positions 105-124 - The fifth title, in
c                                                           integer form
c                                       position 125 - unused
c                                       position 126 - unused
c                                       position 127 - numvdof, number of velocity
c                                                      items per node (ANSYS
c                                                      transient)
c                                       position 128 - numadof, number of
c                                                      acceleration items per
c                                                      node (ANSYS transient)
c                                       position 131-133 - position of velocity
c                                                          in DOF record
c                                                          (ANSYS transient)
c                                       position 134-136 - position of acceleration
c                                                          in DOF record
c                                                          (ANSYS transient)
c                                       position 137-142 - velocity and
c                                                          acceleration labels
c                                                          (ANSYS transient)
c                                       position 143 - number of stress items
c                                                      (6 or 11); a -11 indicates
c                                                      to use principals directly
c                                                      and not recompute (for PSD)
c                                       position 144-146 - position of rotational
c                                                          velocity in DOF record
c                                                          (ANSYS transient)
c                                       position 147-149 - position of rotational
c                                                          accel. in DOF record
c                                                          (ANSYS transient)
c                                       position 150-155 - rotational velocity and
c                                                          acceleration labels
c                                                          (ANSYS transient)
c                                       position 160 - pointer to CINT results
c                                       position 161 - size of CINT record
c                                       position 162 - Pointer to  XFEM/SMART-crack surface information
c                                       position 163 - size of crk surface records
c                                       position 164-200 - unused

c   GEO     ---      1     varies    Entire geometry record for this set of
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                    results data.  Note, when the mesh does not
c                                    change during solution, this pointer will
c                                    simply point to the original GEO record
c                                    stored at the start of the file.  When the
c                                    mesh changes the new geoemtry data will be
c                                    stored here.  The geomID can be used to
c                                    determine when the mesh changes.

c * NSL     dp       1  nnod*Sumdof  The DOF solution for each node in the nodal
c                                    coordinate system.  The DOF order is the
c                                    same as shown above in the DOF number
c                                    reference table.  The nodal order is the
c                                    same order given above in the nodal
c                                    equivalence table.  If a DOF for a node
c                                    isn't valid, a value of 2.0**100 is used.
c                                    Note 1: Sumdof = numdof
c                                    Note 2: If, upon reading of this record,
c                                    there is less than nnod*Sumdof items in the
c                                    record, then only a selected set of nodes
c                                    were output. Another record follows
c                                    (integer, less than nnod long) which
c                                    contains the list of nodes for which DOF
c                                    solutions are available.
c                                      (bit 10 (PDBN) in mask)

c   VSL      dp      1 nnod*numvdof  The velocity solution for each node in the
c                                    nodal coordinate system. The description
c                                    for the DOF solution above also applies
c                                    here.
c                                    ANSYS transient. (bit 27 (PDVEL) in mask)

c   ASL      dp      1 nnod*numadof  The acceleration solution for each node in
c                                    the nodal coordinate system. The
c                                    description for the DOF solution above
c                                    also applies here.
c                                    ANSYS transient. (bit 28 (PDACC) in mask)

c   OSL      dp      1  nnod*Sumdof  extra solution vector for element team
c                                    rezoning project.  To be used later.

c   RF     LONG      1       nrf     Reaction force DOFs.  This index is
c                                    calculated as (N-1)*numdof+DOF, where N is
c                                    the position number of the node in the
c                                    nodal equivalence table, and DOF is the DOF
c                                    reference number.
c                                      (bit 11 (PDBR) in mask)

c * ---     dp       1       nrf     Reaction forces.  The force values are
c                                    ordered according to the DOF order shown
c                                    above in the DOF number reference table.
c                                      (bit 11 (PDBR) in mask)

c   MST    LONG      1       nmast   Master DOF list.  This index is calculated
c                                    as (N-1)*numdof+DOF, where N is the
c                                    position number of the node in the nodal
c                                    equivalence table, and DOF is the DOF
c                                    reference number.

c   BC       i       1        40     Boundary condition index table.
c                                      (bit 23 (PDBBC) in mask)
c                                    numdis,ptrDIX,ptrDIS,numfor,ptrFIX,
c                                    ptrFOR,format,     0,     0,     0,
c                                         0,     0,     0,     0,     0,
c                                         0,     0,     0,     0,     0,
c                                         0,     0,     0,     0,     0,
c                                         0,     0,     0,     0,     0,
c                                         0,     0,     0,     0,     0,
c                                         0,     0,     0,     0,     0

c                                 each item is described below:
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                       numdis - number of nodal constraints
c                                       ptrDIX - pointer to the table of nodes
c                                                having nodal constraints
c                                       ptrDIS - pointer to nodal constraint values
c                                       numfor - number of nodal input force
c                                                loadings
c                                       ptrFIX - pointer to the table of nodes
c                                                having nodal forces
c                                       ptrFOR - pointer to nodal force values
c                                       format - key (0 or 1) denoting which format
c                                                is used for DIX/FIX data (see below)

c                                       positions 7-40 are unused

c   DIX      i       1      numdis      if format == 0 --> Nodal constraint DOF.
c                                       This index is calculated as N*32+DOF,
c                                       where N is the node number and DOF is
c                                       the DOF reference number.  Values are in
c                                       the same order as the DOF number reference
c                                       table.
c
c                                       if format == 1 --> Nodal constraint node
c                                       numbers.

c   ---      i       1      numdis      if format == 0 --> does not exist.
c
c                                       if format == 1 --> Nodal constraint DOF.
c                                       Values are in the same order as the DOF
c                                       number reference table.

c   DIS     dp       1    4*numdis      Nodal constraints.  This record contains
c                                       present and previous values (real and
c                                       imaginary) of the nodal constraints at each
c                                       DOF.

c   FIX      i       1      numfor      if format == 0 --> Nodal input force DOFs.
c                                       This index is calculated as N*32+DOF,
c                                       where N is the node number and DOF is
c                                       the DOF reference number.  Values are in
c                                       the same order as the DOF number reference
c                                       table.
c
c                                       if format == 1 --> Nodal input force node
c                                       numbers.

c   ---      i       1      numfor      if format == 0 --> does not exist.
c
c                                       if format == 1 --> Nodal input force DOF.
c                                       Values are in the same order as the DOF
c                                       number reference table.

c   FOR     dp       1    4*numfor      Nodal forces.  This record contains present
c                                       and previous values (real and imaginary) of
c                                       the nodal input force loadings at each DOF.

c   OND      i       1       nnod       Ordered node list.  This record exists for
c                                       a load case file only.

c   OEL      i       1       nelm       Ordered element list.  This record exists
c                                       for a load case file only.

c   DED      i       1   sizeDead       List of dead elements (EKILL). Uses the
c                                       compressed CMBLOCK format

c   MSHI     i       1      nMSHI       Mulistage harmonic indices for each stage

c   ESL      i       1     2*nelm       Element solutions index table.  This record
c                                       contains pointers to each element solution.
c                                       The order of the elements is the same as
c                                       the order in the element equivalence table.
c                                         (bit 12 (PDBE) in mask)
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

```
c   RCM     dp       1  6*numRotCmp  Angular velocities (3) and angular
c                                     accelerations (3) of components.

c   DMI     dp       1  3+nContours  Crack ID, Contour ID, TipNode, J Integral
c                                     values
c                                       (bit 29 (PDBCINT) in mask)

c       The solution information for each individual element is stored starting
c       at this point in the file.  The next 26 records on the file are
c       repeated as a group nelm times (once for each element).  Item nelm is
c       defined in the file header.

c   ---      i       1       26      Individual element index table.

c                                     ptrEMS, ptrENF, ptrENS, ptrENG, ptrEGR,
c                                     ptrEEL, ptrEPL, ptrECR, ptrETH, ptrEUL,  (10)
c                                     ptrEFX, ptrELF, ptrEMN, ptrECD, ptrENL,
c                                     ptrEHC, ptrEPT, ptrESF, ptrEDI, ptrETB,  (20)
c                                     ptrECT, ptrEXY, ptrEBA, ptrESV, ptrMNL
c                                     ptrESR
c                                    (Relative to ptrESL)

c                                   each item is described below:

c                                     ptrEMS - pointer to misc. data
c                                     ptrENF - pointer to nodal forces
c                                     ptrENS - pointer to nodal stresses
c                                     ptrENG - pointer to volume and energies
c                                     ptrEGR - pointer to nodal gradients
c                                     ptrEEL - pointer to elastic strains
c                                     ptrEPL - pointer to plastic strains
c                                     ptrECR - pointer to creep strains
c                                     ptrETH - pointer to thermal strains
c                                     ptrEUL - pointer to euler angles
c                                     ptrEFX - pointer to nodal fluxes
c                                     ptrELF - pointer to local forces
c                                     ptrEMN - pointer to misc. non-sum values
c                                     ptrECD - pointer to element current
c                                              densities
c                                     ptrENL - pointer to nodal nonlinear data
c                                     ptrEHC - pointer to calculated heat
c                                              generations
c                                     ptrEPT - pointer to element temperatures
c                                     ptrESF - pointer to element surface
c                                              stresses
c                                     ptrEDI - pointer to diffusion strains
c                                     ptrETB - pointer to ETABLE items(post1 only
c                                     ptrECT - pointer to contact data
c                                     ptrEXY - pointer to integration point
c                                              locations
c                                     ptrEBA - pointer to back stresses
c                                     ptrESV - pointer to state variables
c                                     ptrMNL - pointer to material nonlinear record
c                                     ptrESR - pointer to selected results

c                                     Note! If ptrXXX is negative, then all
c                                     |ptrXXX| items are zero and are not on
c                                     the file.

c   EMS     dp       1    varies     Element summable miscellaneous data.  The
c                                     contents and number of data items is
c                                     element-dependent.  For a list of what's
c                                     available, see the SMISC item in the
c                                     description of the ETABLE command in the
c                                     ANSYS Commands Reference.

c   ENF     dp       1    varies     Element nodal forces.  This record contains
c                                     the forces at each node, in the same DOF
c                                     order as the DOF number reference table.
c                                     For static, damping, and inertia forces, a
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                       set of forces will be repeated (as
c                                       appropriate).  Number of data items stored
c                                       in this record can be calculated as
c                                       follows: nodfor*NDOF*M, where NDOF is the
c                                       number of DOFs/node for this element,
c                                       nodfor is the number of nodes per element
c                                       having nodal forces (defined in element
c                                       type description record), and M may be 1,
c                                       2, or 3.  For a static analysis, M=1 only.
c                                       For a transient analysis, M can be 1, 2,
c                                       or 3.

c   ENS     dp       1    varies      Element nodal component stresses.  This
c                                       record contains the stresses at each corner
c                                       node, in the order SX,SY,SZ,SXY,SYZ,SXZ.
c                                       Nodal order corresponds to the connectivity
c                                       defined in the element description.
c                                       Stresses can be nodal values extrapolated
c                                       from the integration points or values at
c                                       the integration points moved to the nodes.
c                                       If an element is nonlinear, integration
c                                       point values always will be written.  (See
c                                       item rxtrap in the solution header for the
c                                       setting.)  An element is considered
c                                       nonlinear when either plastic, creep, or
c                                       swelling strains are present.
c
c                                       Definition of common terms referred here
c                                       and in subsequent EEL, EPL, ECR, ETH,
c                                       ENL, EUL EPT, and EDI sections:
c
c                                       nodstr - number of nodes per element
c                                                having stresses, strains, etc.
c                                                For higher-order elements, nodstr
c                                                equals to the number of corner
c                                                nodes (e.g., for 20-noded SOLID186,
c                                                nodstr = 8). NOTE- the /config NST1
c                                                option may suppress all but one node
c                                                output of nodal ENS EEL EPL ECR ETH
c                                                and ENL for any given element.
c
c                                                NOTE - for layered Prism-shaped SOLID186,
c                                                nodstr = 6
c
c                                       nodfor - number of nodes per element
c                                                having nodal forces, etc.
c                                       ncomp  - number of solution items per node
c                                                ncomp =   6 for ENS record
c                                                          7 for EEL record
c                                                          7 for EPL record
c                                                          7 for ECR record
c                                                          8 for ETH record
c                                                         10 for ENL record
c                                                          7 for EDI record
c                                       NL     - number of layers in layered
c                                                elements
c
c                                       Note: For result sets with NoPrin=0,
c                                       the ENS record will have ncomp=11 and include
c                                       the principal stresses S1,S2,S3,SI,SIGE.
c
c                                       * For solid elements or layered solid elements
c                                         with KEYOPT(8)=0, the record contains
c                                         stresses at each corner node, and
c                                         the number of items in this record is
c                                         nodstr*ncomp.
c                                       * For shell elements or layered shell elements
c                                         with KEYOPT(8)=0, the record contains
c                                         stresses at each corner node (first
c                                         at the bottom shell surface, then the top
c                                         surface), and the number of items in this
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                       record is 2*nodstr*ncomp.
c                                     * For layered elements SHELL91, SHELL99,
c                                       SOLID46, and SOLID191 with KEYOPT(8) = 0,
c                                       if failure criteria were used, the record
c                                       contains additional stresses at each corner
c                                       nodes (first the bottom surface, then the
c                                       top surface) of the layer with the largest
c                                       failure criteria. Therefore, the total number
c                                       of items is 4*nodstr*ncomp for SHELL91 and
c                                       SHELL99, and 2*nodstr*ncomp for SOLID46 and
c                                       SOLID191.
c                                     * For layered elements (with KEYOPT(8)=1),
c                                       stresses for each layer are at each
c                                       corner node (first at the bottom surface, then
c                                       at the top surface), and the number of
c                                       items in this record is NL*2*nodstr*ncomp for
c                                       layered shells and NL*nodstr*ncomp for
c                                       layered solid elements.
c                                     * For layered shell elements with KEYOPT(8)=2,
c                                       the record contains stresses for each layer
c                                       at each corner node  (first at the bottom
c                                       surface, then the top, and finally the middle
c                                       surface). Therefore, the number of items
c                                       in this record is NL*3*nodstr*ncomp.
c                                     * For layered membrane elements (SHELL181,
c                                       SHELL281, SHELL208, and SHELL209 with
c                                       KEYOPT(1)=1 and KEYOPT(8)=1), the record
c                                       contains stresses for each layer at each
c                                       corner node, and the number of items in
c                                       this record  is NL*nodstr*ncomp.
c                                     * For beam elements, the contents and number
c                                       of data items is element-dependent.  See
c                                       the Output Data section for the particular
c                                       element in the ANSYS Elements Reference.
c

c   ENG     dp       1    varies     Element volume and energies.

c                                     * if cpxeng = 0, lrec = 11:
c
c                                       volu,  sene,   aene,   kene,  coene,
c                                     incene,   0.0,    0.0,  thene,    0.0, (10)
c                                        0.0

c                                     * if cpxeng = 1, lrec = 21:
c
c                                       volu,  sene,   aene,   kene,  coene,
c                                     incene,   0.0,    0.0,  thene,    0.0, (10)
c                                        0.0,  dene,   wext,  asene,  akene,
c                                      psene, pkene, asint1, akint1, asint2, (20)
c                                     akint2

c                                    each item is described below:

c                                       volu   - element volume
c                                       sene   - element energy associated with
c                                                the stiffness matrix
c                                       aene   - artificial hourglass energy
c                                       kene   - kinetic energy
c                                       coene  - co-energy (magnetics)
c                                       incene - incremental energy (magnetics)
c                                       0.0    - position not used
c                                       0.0    - position not used
c                                       thene  - thermal dissipation energy
c                                                (see ThermMat, shell131/132 only)
c                                       0.0    - position not used
c                                       0.0    - position not used
c                                       dene   - damping energy
c                                       wext   - work due to external element load
c                                       asene  - amplitude stiffness energy
c                                       akene  - amplitude kinetic energy
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                       psene  - peak stiffness energy
c                                       pkene  - peak kinetic energy
c                                       asint1 - first intermediate result
c                                                for asene calculations
c                                       akint1 - first intermediate result
c                                                for akene calculations
c                                       asint2 - second intermediate result
c                                                for asene calculations
c                                       akint2 - second intermediate result
c                                                for akene calculations

c   EGR     dp      1    varies     Element nodal field gradients.  This record
c                                   contains the gradients at each corner node
c                                   in the order X,Y,Z.  Nodal order
c                                   corresponds to the connectivity defined in
c                                   the element description.  If this is a
c                                   coupled-field analysis, the data is stored
c                                   in the following order (as available):
c                                   fluid, thermal (TEMP), electric (VOLT),
c                                   magnetic (AZ), and diffusion (CONC).
c                                   Gradients can be nodal values extrapolated
c                                   from the integration points or values at the
c                                   integration points moved to the nodes.  See
c                                   item rxtrap in the solution header for the
c                                   setting.  The number of items in this record
c                                   is nodstr*3*N, where N can be 1, 2, 3, or 4
c                                   (depending on the coupled-field
c                                   conditions).

c                                   NOTE: nodstr is defined in the element type
c                                   description record.

c   EEL     dp      1    varies     Element nodal component elastic strains.
c                                   This record contains strains in the order
c                                   X,Y,Z,XY,YZ,XZ,EQV. Elastic strains can be
c                                   can be nodal values extrapolated from the
c                                   integration points or values at the
c                                   integration points moved to the nodes.  If
c                                   an element is nonlinear, integration point
c                                   values always will be written.  See item
c                                   rxtrap in the solution header for the
c                                   setting.  An element is considered
c                                   nonlinear when either plastic, creep, or
c                                   swelling strains are present.  For beam
c                                   elements, see item LEPEL in the description
c                                   in the Output Data section for the
c                                   particular element in the ANSYS Elements
c                                   Reference.
c
c                                   NOTE: See ENS record section for more details
c                                   on record content and length.
c

c   EPL     dp      1    varies     Element nodal component plastic strains.
c                                   This record contains strains in the order
c                                   X,Y,Z,XY,YZ,XZ,EQV.
c                                   Plastic strains are always values at the
c                                   integration points moved to the nodes.  For
c                                   beam elements, see item LEPPL in the
c                                   Output Data section for the particular
c                                   element in the ANSYS Elements Reference.
c
c                                   NOTE: See ENS record section for more details
c                                   on record content and length.
c

c   ECR     dp      1    varies     Element nodal component creep strains.
c                                   This record contains strains in the order
c                                   X,Y,Z,XY,YZ,XZ,EQV.
c                                   Creep strains are always values at the
c                                   integration points moved to the nodes.  For
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                       beam elements, see item LEPCR in the
c                                       Output Data section for the particular
c                                       element in the ANSYS Elements Reference.
c
c                                       NOTE: See ENS record section for more details
c                                       on record content and length.
c

c   ETH     dp       1    varies      Element nodal component thermal strains.
c                                       This record contains strains in the order
c                                       X,Y,Z,XY,YZ,XZ,EQV plus the element
c                                       swelling strain. Thermal
c                                       strains can be nodal values extrapolated
c                                       from the integration points or values at
c                                       the integration points moved to the nodes.
c                                       If the element in nonlinear, integration
c                                       point data always will be written. (An
c                                       element is considered nonlinear when either
c                                       plastic, creep, or swelling strains are
c                                       present.) See item rxtrap in the solution
c                                       header for the setting.  For beam elements,
c                                       see item LEPTH in the description of the
c                                       Output Data section for the particular
c                                       element in the ANSYS Elements Reference.
c
c                                       NOTE: See ENS record section for more details
c                                       on record content and length.
c

c   EUL     dp       1    varies      Element Euler angles.  This record contains
c                                       the Euler angles (THXY,THYZ,THZX). (No
c                                       attempt is made to make this information
c                                       complete for all cases of all element
c                                       types.  Programmers need to verify their
c                                       situations.)
c
c                                       ** FOR OLDER ELEMENTS **
c                                       --For lower-order elements
c                                         (e.g. PLANE42 and SOLID45), angles are
c                                         at the centroid and the number of items
c                                         in this record is 3.
c                                       --For higher-order elements (e.g.
c                                         PLANE82, SOLID92, and SOLID95), angles
c                                         are at each corner node and the number of
c                                         items in this record is nodstr*3.
c                                         The above two categories are output if
c                                         ESYS is used, a material KEYOPT is used
c                                         (e.g. KEYOPT(1) for PLANE42), or if it is
c                                         from a large deflection superelement
c                                         stress pass
c
c                                       ** FOR NEW GENERATION SHELL ELEMENTS **
c                                       --For SHELL181/SHELL281:
c                                          For real constant input: the number of
c                                            items in this record is 12.
c                                          For section input:
c                                           IF KEYOPT(8) > 0:
c                                            the number of items in this record is
c                                                            12 + number of layers
c                                           IF KEYOPT(8) = 0:
c                                             IF regular shell  (KEYOPT(1) = 0)
c                                                the number of items in this record
c                                                 is 14
c                                             IF membrane shell (KEYOPT(1) = 1):
c                                               IF number of layers = 1, the number
c                                                 of items in this record is 13
c                                               IF number of layers > 1, the number
c                                                 of items in this record is 14
c
c                                       ** FOR THE NEW GENERATION SOLID ELEMENTS **
c                                       --For uniform reduced integration lower-order
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                       elements (e.g. PLANE182, KEYOPT(1)=1 and
c                                       SOLID185 KEYOPT(2)=1):
c                                       the angles are at the centroid and the number
c                                       of items is 3.
c                                     --For other formulations of lower-order
c                                       elements (e.g. PLANE182 and SOLID185) and
c                                       the higher-order elements
c                                       (e.g. PLANE183, SOLID186, and SOLID187):
c                                       The number of items in this record is
c                                       (nodstr*3).
c                                     --For layered solid elements, add NL values,
c                                       so that the number of items in this record
c                                       is (nodstr*3)+NL.
c
c                                     NOTE: See ENS record section for definition of
c                                           terms NL and nodstr.

c   EFX     dp       1    varies     Element nodal field fluxes.  This record
c                                    contains the fluxes at each corner node in
c                                    the order X,Y,Z. If this is a
c                                    coupled-field analysis, the flux data is
c                                    stored in the following order:  thermal,
c                                    electric, magnetic.  Nodal order
c                                    corresponds to the connectivity defined in
c                                    the element description.  Fluxes can be
c                                    nodal values extrapolated from the
c                                    integration points or values at the
c                                    integration points moved to the nodes.
c                                    See item rxtrap in the solution header for
c                                    the setting.  The number of items in this
c                                    record is nodstr*3*N, where N can be 1, 2,
c                                    or 3 depending on the coupled-field
c                                    conditions.

c                                    NOTE: nodstr is defined in the element type
c                                    description record.

c * ELF     dp       1    varies     Element nodal coupled-field forces.  This
c                                    record lists the forces at each node in the
c                                    order X,Y,Z.  For most elements, the number
c                                    of items in this record is nodfor*3.
c                                    However, for the PLANE53 element, the
c                                    number of items in this record is either
c                                    nodfor*3 or nodstr*3.  (See the description
c                                    of KEYOPT(7) for PLANE53 in the ANSYS
c                                    Elements Reference.)  NOTE: nodfor and
c                                    nodstr are defined in the element type
c                                    description record.

c   EMN     dp       1    varies     Element nonsummable miscellaneous data.
c                                    The contents and number data items for this
c                                    record is element-dependent.   See the
c                                    description for item NMISC of the ETABLE
c                                    command in the ANSYS Commands Reference.

c * ECD     dp       1       3       Element current densities.  This record
c                                    contains the calculated current densities
c                                    in the order X,Y,Z.

c   ENL     dp       1    varies     Element nodal nonlinear data.  This record
c                                    stores nonlinear data at each corner node
c                                    in the order SEPL, SRAT, HPRES, EPEQ,
c                                    PSV or CREQ, PLWK, CRWK, and ELENG
c                                    followed by 2 spares.

c                                   each item is described below:
c                                     SEPL  - equivalent stress parameter
c                                     SRAT  - stress ratio
c                                     HPRES - hydrostatic pressure
c                                     EPEQ  - accumulated equivalent plastic
c                                             strain
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                       PSV   - plastic state variable
c                                       CREQ  - accumulated equivalent creep
c                                               strain.  Applies to current
c                                               technology element types
c                                               180,181,182,183,185,186,
c                                               187,188,189,208,209,265,
c                                               281,288,289,290
c                                       PLWK  - plastic strain energy density(work)
c                                       CRWK  - creep strain energy density (work)
c                                       ELENG - elestic strain energy density

c                                       * See ENS record section for details on
c                                         solid and shell elements.
c                                       * For beam elements, the contents and
c                                         number of data items in this record is
c                                         element-dependent.  See the description
c                                         of item NLIN in the Output Data section
c                                         for the particular element in the ANSYS
c                                         Elements Reference.

c * EHC     dp       1       1       Element heat generation.  This record
c                                    stores the calculated heat generation.

c   EPT     dp       1    varies     Element structural nodal temperatures.

c                                    * For solid elements and SHELL41, the
c                                      record contains nodal temperatures at
c                                      each node and the number of items in this
c                                      record is nodfor.
c                                    * For shell elements, except SHELL41 and
c                                      SHELL91, the record contains nodal
c                                      temperatures at each corner node for the
c                                      top surface and the bottom surface.  The
c                                      number of items in this record is
c                                      nodstr*2.
c                                    * For SHELL91 and SOLID191, the record
c                                      contains nodal temperatures at each
c                                      corner node for the bottom of the bottom
c                                      layer, and each succeeding interlayer
c                                      surface up to the top of the top layer.
c                                      The number of items in this record is
c                                      (NL+1)*nodstr.
c                                    * For layered shell elements SHELL181,
c                                      SHELL281, SHELL208, SHELL209, and layered
c                                      solid elements SOLID185, SOLID186,
c                                      and SOLSH190, the record contains
c                                      temperatures for each layer at each
c                                      corner node (first at the bottom layer
c                                      surface, then the top). Therefore, the number
c                                      of items in this record is NL*2*nodstr for
c                                      layered shells and NL*nodstr for layered
c                                      solid elements.
c                                    * For layered membrane elements (SHELL181,
c                                      SHELL281, SHELL208, and SHELL209 with
c                                      KEYOPT(1)=1), the record contains
c                                      temperatures for each layer at each
c                                      corner node. Therefore, the number of items
c                                      in this record is NL*nodstr.
c                                    * For beam elements, the contents and
c                                      number of data items in this record is
c                                      element-dependent.  See the description
c                                      of item LBFE in the Output Data section
c                                      for the particular element in the ANSYS
c                                      Elements Reference.

c                                    NOTE: See ENS record section for definition
c                                          of terms NL, nodstr, and nodfor.

c   ECT     dp       1    varies     Contact element results.  This record
c                                    stores contact results at each corner node
c                                    in the order STAT, PENE, PRES, SFRIC, STOT,
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                     SLIDE, GAP, FLUX, CNOS, FPRS
c                                   each item is described below:
c                                     STAT  - Contact status
c                                     PENE  - Contact penetration
c                                     PRES  - Contact pressure
c                                     SFRIC - Contact friction stress
c                                     STOT  - Contact total stress (pressure plus friction)
c                                     SLIDE - Contact sliding distance
c                                     GAP   - Contact gap distance
c                                     FLUX  - Total heat flux at contact surface
c                                     CNOS  - Total number of contact status changes during substep
c                                     FPRS  - Actual applied fluid penetration pressure

c   ESF     dp       1    nsurf*19    Element surface stresses.  The
c                                     length of this record is nsurf*19 where
c                                     nsurf is the number of surfaces that have
c                                     surface stress information.  The stress
c                                     information is simply repeated in the
c                                     format shown below for each surface.

c                                     * For 2d elements:

c                                     facenm,  area,  temp, press, eppar,
c                                      epper,   epz, 0.0d0,  spar,  sper,
c                                         sz, 0.0d0, 0.0d0, 0.0d0,    s1,
c                                         s2,    s3,  sint,  seqv

c                                     * For 3d elements:

c                                     facenm,  area,  temp, press,   epx,
c                                        epy,   epz,  epxy,    sx,    sy,
c                                         sz,   sxy, 0.0d0, 0.0d0,    s1,
c                                         s2,    s3,  sint,  seqv

c                                     * For axisymmetric elements:

c                                     facenm,  area,  temp, press, eppar,
c                                      epper,   epz,  epsh,  spar,  sper,
c                                         sz, 0.0d0, 0.0d0,   ssh,    s1,
c                                         s2,    s3,  sint,  seqv

c                                   each item is described below:

c                                     facenm - face number
c                                     area   - face area
c                                     temp   - face temperature
c                                     press  - face pressure
c                                     epx    - strain parallel to face
c                                     epy    - strain parallel to face
c                                     epz    - strain perpendicular to face
c                                     epxy   - shear strain
c                                     eppar  - strain parallel to face
c                                     epper  - strain perpendicular to face
c                                     epsh   - torsion shear strain
c                                     sx     - stress parallel to face
c                                     sy     - stress parallel to face
c                                     sz     - stress perpendicular to face
c                                     sxy    - shear stress
c                                     spar   - stress parallel to face
c                                     sper   - stress perpendicular to face
c                                     ssh    - torsion shear stress
c                                     s1     - S(1)
c                                     s2     - S(2)
c                                     s3     - S(3)
c                                     sint   - S(INT)
c                                     seqv   - S(EQV)
c                                     0.0d0  - position not used

c   EDI     dp       1    varies      Element nodal component diffusion strains.
c                                     This record contains strains in the order
c                                     X,Y,Z,XY,YZ,XZ,EQV. Diffusion
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                       strains can be nodal values extrapolated
c                                       from the integration points or values at
c                                       the integration points moved to the nodes.
c                                       See item rxtrap in the solution header for
c                                       the setting.
c
c                                       NOTE: See ENS record section for more details
c                                       on record content and length.
c

c   EXY     dp       1    varies     Element integration point coordinates
c                                       The length of the record is numint*3, where
c                                       numint is the number of integration points.
c                                       Even two-dimensional elements use the 3.
c                                       They are output only if requested with the
c                                       OUTRES,loci command.
c                                       Applicable only to legacy element types
c                                       2,42,45,82,92,95, and current technology
c                                       element types 180,181,182,183,185,186,187,
c                                       188,189,208,209,265,281,288,289,290

c   EBA     dp       1    varies     Element structural nodal back stresses
c                                       Record has the same form as the plastic
c                                       strains.  They are output if the form of
c                                       plasticity is kinematic hardening and the
c                                       plastic strains are requested.
c                                       Applicable only to legacy element types
c                                       2,42,45,82,92,95, and current technology
c                                       element types 180,181,182,183,185,186,187,
c                                       188,189,208,209,265,281,288,289,290

c   ESV     dp       1    varies     Element state variable record.  Exists only
c                                       if written by user in usermat or usercreep.

c   MNL     dp       1    varies     Material nonlinear record.

c   ESR     dp       1    varies     Element nodal selected results. The length
c                                       is NINT(ESR(1)). The record includes
c                                       type specifications and results values.
c                                       The type of results depends on the selected
c                                       result output specfications.

c   records marked with * to the left of the record id can be read and stored
c   into database with "ldread" command.


c        The solution information for the Nodal Averaged Results (NAR) is stored
c        starting at this point in the file.

c   NAR      i       1       10      NAR index table.

c                                       ptrNCTl, ptrNCTh, ptrNSTl, ptrNSTh, ptrNELl,
c                                       ptrNELh, ptrNPLl, ptrNPLh, ptrNCRl, ptrNCRh, (10)
c                                       ptrNTHl, ptrNTHh
c                                      (Relative to ptrNAR)

c                                       NOTE: nodal results are stored in the same
c                                             node order as given above in the nodal
c                                             equivalence table.  If a DOF for a
c                                             node isn't valid, a value of 2.0**100
c                                             is used.

c   NCT      i       1    4*nnod     Node Contributions. This record contains
c                                       the number of elements that shared each node
c                                       and contributed to the calculated value for
c                                       each NAR vector. The first nnod values are
c                                       the node contributions for the nodal
c                                       stresses, the next nnod values are for the
c                                       node contibutions for nodal elastic
c                                       strains, etc.
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c   NST     dp       1    6*nnod     Nodal component stresses.
c                                    This record contains the stresses at each
c                                    node, in the order X,Y,Z,XY,YZ,XZ.

c   NEL     dp       1    7*nnod     Nodal component elastic strains.
c                                    This record contains the strains at each
c                                    node, in the order X,Y,Z,XY,YZ,XZ,EQV.

c   NPL     dp       1    7*nnod     Nodal component plastic strains.
c                                    This record contains the strains at each
c                                    node, in the order X,Y,Z,XY,YZ,XZ,EQV.

c   NCR     dp       1    7*nnod     Nodal component creep strains.
c                                    This record contains the strains at each
c                                    node, in the order X,Y,Z,XY,YZ,XZ,EQV.

c   NTH     dp       1    8*nnod     Nodal component thermal and swelling
c                                    strains. This record contains the strains
c                                    at each node, in the order X,Y,Z,XY,YZ,XZ,
c                                    EQV plus the node swelling strain


c *** Nodal Component Stresses (unused)

c  NDSTR    dp       1    6*nnod     Nodal component stresses (TOP for shells)
c                                     (nNodStr > 0)
c           dp       1    6*nnod     BOT nodal component stresses for shells
c                                     (nNodStr > 1)
c           dp       1    6*nnod     MID nodal component stresses for shells
c                                     (nNodStr > 2)
```

1.3. Description of the Reduced Displacement File

This section explains the content of the reduced displacement file (`jobname.rdsp`).

1.3.1. Standard Mechanical APDL File Header

See The Standard Header for Mechanical APDL Binary Files (p. 4) for a description of this set. File number (Item 1) is 10.

1.3.2. RDSP File Format

```
*comdeck,fdrdsp

c *** Copyright ANSYS.  All Rights Reserved.
c *** ansys, inc


c     **********  description of reduced displacement file  **********
      character*8  RDSPNM
      parameter  (RDSPNM='rdsp    ')

      integer         RDSPHDLEN
      parameter      (RDSPHDLEN=80)

      LONGINT         rdspfpL, rdspfp
      integer         rdspbk,  rdsput
      common /fdrdsp/ rdspfpL, rdspbk, rdsput
      equivalence  (rdspfp,rdspfpL)

c  write:  lnfrcl,lnfrin,lnfrwr
c  read:   rdtrs
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c   ********** common variable descriptions ***********
co  rdspfpL      file position on file rdsp
co  rdspbk       block number for file rdsp
co  rdsput       file unit for file rdsp

c   See fddesc for documentation of how binary files are stored.
c
c     **********  file format  **********

c        recid tells the identifier for this record.  Not all records will have
c             identifiers -- they are only indicated for those records whose
c             record pointers are stored in the second file header.

c        type tells what kind of information is stored in this record:
c             i - integer
c             dp - double precision
c             cmp - complex

c        nrec tells how many records of this description are found here

c        lrec tells how long the records are (how many items are stored)

c recid    type    nrec    lrec     contents

c  ---      i       1      100      standard ANSYS file header (see binhed8 for
c                                   details of header contents)

c  ---      i       1       80      .RDSP FILE HEADER
c
c                                fun10,    nmrow,   nmatrx,    nmode,   numdof,
c                                 maxn,    wfmax,   lenbac,    ngaps,   ncumit, (10)
c                                   kan,     nres,     ndva,    nvect,   DSPfmt,
c                                minmod,        0,  modlstp, ndefdval,  nEnfDof, (20)
c                                ptrDOF,  ptrDAMP, ptrDAMPh,        0,        0,
c                                ptrFRQ,   ptrDSP,        0,        0,        0, (30)
c                               ptrFRQh,  ptrDSPh,   ptrDVA,  ptrDVAh,nrkeyPert,
c                              kPerturb,    keyVA,Glblenbac,        0,        0, (40)
c                                     0,        0,        0,        0,        0,
c                                     0,        0,        0,        0,        0, (50)
c                                     0,        0,        0,        0,        0,
c                                     0,        0,        0,        0,        0, (60)
c                                     0,        0,        0,        0,        0,
c                                     0,        0,        0,        0,        0, (70)
c                                     0,        0,        0,        0,        0,
c                                     0,        0,        0,        0,        0  (80)

c                                  each item in header is described below:

c                                    fun10  - unit number (rdsp file is 10)
c                                    nmrow  - number of rows/columns in matrices
c                                    nmatrx - number of reduced matrices on the
c                                             file
c                                    nmode  - number of modes extracted during
c                                             modal analysis
c                                    numdof - number of dofs per node
c                                    maxn   - maximum node number
c                                    wfmax  - maximum wavefront
c                                    lenbac - number of nodes
c                                    ngaps  - number of gaps
c                                    ncumit - total number of iterations done
c                                             during analysis
c                                    kan    - analysis type
c                                             = 5 for MSUP transient analysis
c                                    nres   - number of residual vectors used
c                                    modlstp- multiple load step key
c                                    ndva   - length of DVA (for restart)
c                                    nvect  - number of available load vectors
c                                    DSPfmt - 0,physical disps  .ne.0,modal coords
c                                    minmod - smallest mode number used
c                                    ndefdval- number of defined enforced motion
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                  maxEnf - maximum enforced values
c                                  ptrDOF - pointer to degree of freedom set
c                                  ptrDAMP  - pointer to damping values
c                                  ptrDAMPh - High part of DAMP pointer
c                                  ptrDNC - pointer to nodal constraints
c                                  ptrFRQ - pointer to the frequencies
c                                  ptrDSP - pointer to the step solution data:
c                                              - calculated displacements
c                                              - load vector scale factors
c                                              - gap restoring forces
c                                              - calculated velocities
c                                              - calculated accelerations
c                                  ptrFRQh- High part of frequency ptr
c                                  ptrDSPh- High part of displacement ptr
c                                  ptrDVA - pointer to modal disp, velo and acc (for restart)
c                                  ptrDVAh- High part of modal disp, velo and acc (for restart)
c                                  nrkeyPert - nrkey setting of base analysis (Linear Perturbation)
c                                  kPerturb - Linear Perturbation key
c                                  keyVA  - Key for velocities/accelerations on file
c                                  Glblenbac - global number of nodes (== 0 unless using Distributed Ansys)
c                                  0      - position not used

c  ---      i       1     numdof   Degrees of freedom per node
c                                   (curdof(i),i=1,numdof)
c                                  dof reference numbers are:
c         UX  = 1, UY  = 2, UZ  = 3, ROTX= 4, ROTY= 5, ROTZ= 6, AX  = 7, AY  = 8
c         AZ  = 9, VX  =10, VY  =11, VZ  =12, GFV1=13, GFV2=14, GFV3=15, WARP=16
c         CONC=17, HDSP=18, PRES=19, TEMP=20, VOLT=21, MAG =22, ENKE=23, ENDS=24
c         EMF =25, CURR=26, SP01=27, SP02=28, SP03=29, SP04=30, SP05=31, SP06=32

c  ---      i       1     lenbac  Nodal equivalence table. This table equates
c                                   the number used for storage to the actual
c                                   node number
c                                  (Back(i),i=1,lenbac)

c  ---      i       1  Glblenbac  Global nodal equivalence table.  This
c                                   table equates the number used for storage
c                                   to the actual node number.  Only written
c                                   by the master process in Distributed Ansys
c                                  (GlbBack(i),i=1,Glblenbac)

c  ---      dp      1       10    Time information:
c                                   dtime,    0.0,    0.0,    0.0,    0.0,
c                                     0.0,    0.0,    0.0,    0.0, timend

c                                 each item is described below:
c
c                                   dtime  - the time increment
c                                   timend - the final time of the analysis
c                                   0.0    - position not used

c  DOF      i       1     nmrow    Degree of freedom set used
c                                   The DOFs are calculated as (N-1)*numdof+DOF,
c                                   where N is the position number of the node in
c                                   the nodal equivalence table and DOF is the
c                                   DOF reference number given above.

c  ---      i       1     nmrow+1  Original reduced set of DOFs used.
c                                   The DOFs are calculated as (N-1)*numdof+DOF,
c                                   where N is the position number of the node in
c                                   the nodal equivalence table and DOF is the
c                                   DOF reference number given above.

c  DAMP     dp      1   nmode+10   Damping values.
c                                   This record is present only for analysis using
c                                   the mode superposition method with MCkey
c                                   activated - TRNOPT cmd.
c                                   There are nmode+10 entries:
c                                   1 : nmode        - effective damping ratios
c                                   nmode+1 : nmode+5  - alphad, betad, dmprat, dmpstr, dsmpFreq
c                                   nmode+6 : nmode+10 - additional entries - zeroed out for now
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

```
c  FRQ      dp      1     nmrow/   Frequencies extracted from the modal analysis.
c                        ndva/    This record is present only for analyses using
c                     ndva-nEnfDof the mode superposition method.
c                                  If DSPfmt is 0, the first nmrow values are the
c                                  frequencies extracted from the modal analysis.
c                                  The remaining values have no meaning. Otherwise,
c                                  there are ndva frequencies if there are no
c                                  enforced motions and ndva less the number of enforcced
c                                  motions otherwise.
c                                  If DSPfmt=0 :(freq(i),i=1,nmrow)
c                                  If DSPfmt!=0 and ndefdval=0:
c                                    (freq(i),i=1,ndva)
c                                  If DSPfmt!=0 and ndefdval>0:
c                                    (freq(i),i=1,ndva-nEnfDof)

c  *** The next 6 to 7 records are repeated (as a group) until the time value
c  *** equals the value of timend.  The number of iterations is stored as
c  *** ncumit. (see above records that deal with time)

c  DSP      dp      1     nmrow+7/ Calculated displacements
c                        ndva+7   The first nmrow entries are the displacements
c                                  in the same order as the original set of DOFs
c                                  (see record AFTER ptrDOF). If DSPfmt=0, these
c                                  are physical displacements, If DSPfmt!=0,
c                                  these are the ndva modal coordinates instead
c                                  of the nmrow entries.
c
c                                  For the last six entries:
c                                   1. Time for these displacements
c                                   2. Load step number
c                                   3. Substep number
c                                   4. Cumulative iteration number
c                                   5. Scale factor
c                                   6. numdeflvs - number of scale factors
c                                   7. kwrval - key to control writing velocities
c                                             and accelerations at each substep (write if keyVA = 1 and kwrval =
c
c  ---      i       1             Note: If, upon reading of this record, there
c                                  is less than nmrow+5 items in the record,
c                                  then only a selected set of nodes were
c                                  output. Another record follows (integer, less
c                                  than lenbac long) which contains the list of
c                                  nodes for which DOF solutions are available.

c  ---      i       1 numdeflvs    lvscal table scale factor IDs
c                                   (ilvscID(i),i=1,numdeflvs)

c  ---      dp      1 numdeflvs    lvscal table scale factor values
c                                   (dlvscVal(i),i=1,numdeflvs)

c  ---      dp      1     ngaps    Gap restoring forces.  This record is
c                                  present only if ngaps > 0.
c                                   (fgaps(i),i=1,ngaps)

c  ---      dp      1 nmrow/ndva   Calculated velocities (present if keyVA = 1).
c                                  nmrow entries if DSPfmt=0, ndva entries if DSPfmt!=0

c  ---      dp      1 nmrow/ndva   Calculated accelerations (present if keyVA = 1)
c                                  nmrow entries if DSPfmt=0, ndva entries if DSPfmt!=0

c *** The next 3 records are kept for possible restart in mode superposition
c *** transient. They are overwritten upon restarting. They are written once (last
c *** loadstep).

c  DVA      dp      1     ndva+6  Calculated modal displacements
c                                   The first ndva entries are the modal
c                                   displacements. For the last six entries:
c                                    1. Time for these displacements
c                                    2. Load step number
c                                    3. Substep number
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                  4. Cumulative iteration number
c                                  5. Scale factor
c                                  6. numdeflvs - number of scale factors

c  ---      dp      1     ndva    Calculated modal velocities

c  ---      dp      1     ndva    Calculated modal accelerations
```

1.4. Description of the Reduced Complex Displacement File

This section explains the content of the reduced complex displacement file (`jobname.rfrq`).

1.4.1. Standard Mechanical APDL File Header

See The Standard Header for Mechanical APDL Binary Files (p. 4) for a description of this set. File number (Item 1) is 10.

1.4.2. RFRQ File Format

```
*comdeck,fdrfrq

c *** Copyright ANSYS.  All Rights Reserved.
c *** ansys, inc


c     **********  description of reduced complex displacement file  **********
      character*8  RFRQNM
      parameter  (RFRQNM='rfrq    ')

      LONGINT         rfrqfpL, rfrqfp
      integer         rfrqbk,  rfrqut
      common /fdrfrq/ rfrqfpL, rfrqbk, rfrqut
      equivalence  (rfrqfp,rfrqfpL)

c   ********** common variable descriptions ***********
co  rfrqfpL      file position on file rfrq
co  rfrqbk       block number for file rfrq
co  rfrqut       file unit for file rfrq

c   See fddesc for documentation of how binary files are stored.
c
c     **********  file format  **********

c        recid tells the identifier for this record.  Not all records will have
c             identifiers -- they are only indicated for those records whose
c             record pointers are stored in the second file header.

c        type tells what kind of information is stored in this record:
c             i - integer
c             dp - double precision
c             cmp - complex

c        nrec tells how many records of this description are found here

c        lrec tells how long the records are (how many items are stored)

c recid    type    nrec    lrec     contents

c  ---      i       1      100      standard ANSYS file header (see binhed8 for
c                                   details of header contents)

c  ---      i       1       40    .RFRQ FILE HEADER
c
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                    fun10,    nmrow,   nmatrx,    nmode,   numdof,
c                                     maxn,    wfmax,   lenbac,   extopt,   ncumit, (10)
c                                      kan,     nres,   nmUsed,    nvect,   DSPfmt,
c                                   minmod,        0,  modlstp,        0,  nEnfdof, (20)
c                                   ptrDOF,  ptrDAMP, ptrDAMPh,        0,        0,
c                                   ptrFRQ,   ptrDSP,        0,        0,        0, (30)
c                                  ptrFRQh,  ptrDSPh,nrkeyPert,  kPertrb,Glblenbac,
c                                   cpxmod,   SvCode,QRdampKey,        0,        0  (40)

c                                 each item in header is described below:

c                                   fun10  - unit number (rfrq file is 10)
c                                   nmrow  - number of rows/columns in matrices
c                                   nmatrx - number of reduced matrices on file
c                                   nmode  - number of modes extracted during
c                                            modal analysis
c                                   numdof - number of dofs per node
c                                   maxn   - maximum node number
c                                   wfmax  - maximum wavefront
c                                   lenbac - number of nodes
c                                   extopt - mode extraction method
c                                             = 3 - unsymmetric Lanczos
c                                             = 4 - damped Lanczos
c                                             = 6 - block Lanczos
c                                             = 8 - SuperNode
c                                             = 9 - PCG Lanczos
c                                   ncumit - total number of iterations done
c                                            during analysis
c                                   kan    - analysis type
c                                            = 6  - MSUP harmonic analysis
c                                   nmUsed - number of modes used in mode
c                                            superposition
c                                   nvect  - number of generated loads in .mlv
c                                   DSPfmt - 0,physical disps  .ne.0,modal coords
c                                   minmod - smallest mode number used
c                                   modlstp- multiple load step key
c                                   ptrDOF - pointer to degree of freedom set
c                                            used in model
c                                   ptrDAMP  - pointer to damping values
c                                   ptrDAMPh - High part of DAMP pointer
c                                   ptrFRQ - pointer to the frequencies
c                                   ptrDSP - pointer to the calculated
c                                            displacements
c                                   ptrFRQh- High part of FRQ pointer
c                                   ptrDSPh- High part of DSP pointer
c                                   nrkeyPert - nrkey setting of base analysis (Linear Perturbation)
c                                   kPertrb- Linear Perturbation key
c                                   Glblenbac - global number of nodes (== 0 unless using Distributed Ansys)
c                                   cpxmod   - key if complex modes were used
c                                   SvCode - Solver assembly code
c                                            = 1 Symbolic assembly
c                                   QRdampKey - Key for QRDAMP solver use
c                                               for modal analysis preceeding MSUP (MODOPT,QRDAMP)
c                                             = 1 if QRDAMP was used, 0 otherwise
c
c                                   0      - position not used

c  ---      i       1     numdof   Degrees of freedom per node
c                                    (curdof(i),i=1,numdof)
c                                   dof reference numbers are:
c         UX  = 1, UY  = 2, UZ  = 3, ROTX= 4, ROTY= 5, ROTZ= 6, AX  = 7, AY  = 8
c         AZ  = 9, VX  =10, VY  =11, VZ  =12, GFV1=13, GFV2=14, GFV3=15, WARP=16
c         CONC=17, HDSP=18, PRES=19, TEMP=20, VOLT=21, MAG =22, ENKE=23, ENDS=24
c         EMF =25, CURR=26, SP01=27, SP02=28, SP03=29, SP04=30, SP05=31, SP06=32

c  ---      i       1     lenbac  Nodal equivalence table. This table equates
c                                    the number used for storage to the actual
c                                    node number
c                                   (Back(i),i=1,lenbac)

c  ---      i       1  Glblenbac  Global nodal equivalence table.  This
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                    table equates the number used for storage
c                                    to the actual node number.  Only written
c                                    by the master process in Distributed Ansys
c                                   (GlbBack(i),i=1,Glblenbac)

c  ---      dp      1       10     Unused record. contents:
c                                  1.0, 0.0, 0.0, 0.0, 0.0,
c                                  0.0, 0.0, 0.0, 0.0, 0.0

c  DOF      i       1     nmrow    Degree of freedom set used
c                                   The DOFs are calculated as (N-1)*numdof+DOF,
c                                   where N is the position number of the node in
c                                   the nodal equivalence table and DOF is the
c                                   DOF reference number given above.

c  ---      i       1     nmrow+1  Original reduced set of DOFs used.
c                                   The DOFs are calculated as (N-1)*numdof+DOF,
c                                   where N is the position number of the node in
c                                   the nodal equivalence table and DOF is the
c                                   DOF reference number given above.

c  DAMP     dp      1   nmode+10   Damping values.
c                                   This record is present only for analyses using
c                                   the mode superposition method with MCkey
c                                   activated - HROPT cmd.
c                                   There are nmode+10 entries:
c                                   1 : nmode    - effective damping ratios
c                                   nmode+1 : nmode+4  - alphad, alphad, dmprat, dmpstr
c                                   nmode+5 : nmode+10 - additional entries - zeroed out for now

c  FRQ      dp      1     nmrow/   Frequencies extracted from the modal analysis.
c                      nmode+nres  This record is present only for analyses using
c                                  the mode superposition method.
c                                  There are nmrrow entries if DSPfmt=0 and nmode+nres
c                                  entries otherwise.
c                                  If cpxmod
c                                  If DSPfmt=0  : (freq(i),i=1,nmrow)
c                                  If DSPfmt!=0 : (freq(i),i=1,nmode+nres)
c                                  Frequencies are complex if DSPfmt=0 and extopt=3 or 4
c                                  or if DSPfmt!=0 and cpxmod = 1.
c                                  For complex frequencies the number of entries is doubled.
c                                  Real parts appear for first half and imaginary parts next.
c                                  If DSPfmt!=0 and cpxmod = 1:
c                                  (dreal(freq(i)),i=1,nmode+nres)
c                                  (dimag(freq(i)),i=nmode+nres,2*(nmode+nres))

c  *** The next 3 records are repeated (as a pair)
c  *** The number of iterations is stored as ncumit.

c  DSP      cmp  ncumit  nmrow+5/  Calculated complex displacements
c                        nmUsed+5  The first nmrow entries are the displacements
c                                  in the same order as the original set of DOFs
c                                  (see record AFTER ptrDOF). If DSPfmt=0, these
c                                  are physical displacements, If DSPfmt!=0,
c                                  these are the nmUsed modal coordinates instead
c                                  of the nmrow entries.
c
c                                  For the last five entries:
c                                     Real part             Imag part
c                                  1. frequency for these   frequency increment
c                                     values
c                                  2. load step number      substep number
c                                  3. cumulative iteration  emrpm (MRPM cmd)
c                                     number
c                                  4. zero                  zero
c                                  5. scale factor          numdeflvs
c                                       (cvs(i),i=1,nmrow/nmUsed),(freq,delf),
c                                       (itime,itter),(ncumit,0.0),(0.0,0.0),
c                                       (fscale,numdeflvs)

c                                  Note: If, upon reading of this record, there
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                  is less than nmrow+5 items in the record,
c                                  then only a selected set of nodes were
c                                  output. Another record follows (integer, less
c                                  than lenbac long) which contains the list of
c                                  nodes for which DOF solutions are available.

c  ---      i       1 numdeflvs    lvscal table scale factor IDs
c                                   (ilvscID(i),i=1,numdeflvs)

c  ---      dp      1 numdeflvs    lvscal table scale factor values
c                                   (dlvscVal(i),i=1,numdeflvs)
```

1.5. Description of the Modal Results File

This section explains the content of the modal results file (`jobname.mode`).

1.5.1. Standard Mechanical APDL File Header

See The Standard Header for Mechanical APDL Binary Files (p. 4) for a description of this set. File number (Item 1) is 9.

1.5.2. MODE File Format

```
*comdeck,fdmode

c *** Copyright ANSYS.  All Rights Reserved.
c *** ansys, inc.

c     **********  description of modal result file  **********

      character*8  MODENM
      parameter   (MODENM='mode    ')

      character*8  MODENM_LEFT
      parameter   (MODENM_LEFT='lmode   ')

c     *** NOTE: if this variable is changed in the future it should be
c     ***       updated in spdefines.h also for symbolic assembly (jrb)
      integer         MODEHDLEN
      parameter      (MODEHDLEN=100)

      LONGINT         modefpL
      integer         modebk, modeut

      LONGINT         modeLeftfpL
      integer         modeLeftbk, modeLeftut

      common /fdmode/ modefpL, modebk, modeut,
     x                modeLeftfpL, modeLeftbk, modeLeftut

c   ********** common variable descriptions ***********
co  modefpL      file position on file mode
co  modebk       block number for file mode
co  modeut       file unit for file mode
co  modeLeftfpL  file position on file .lmode
co  modeLeftbk   block number for file .lmode
co  modeLeftut   file unit for file .lmode


c   See fddesc for documentation of how binary files are stored.
c
c     **********  file format  **********
```

```
c        recid tells the identifier for this record.  Not all records will have
c             identifiers -- they are only indicated for those records whose
c             record pointers are stored in the second file header.

c        type tells what kind of information is stored in this record:
c             i - integer
c             dp - double precision
c             cmp - complex

c        nrec tells how many records of this description are found here

c        lrec tells how long the records are (how many items are stored)

c recid    type    nrec    lrec     contents

c  ---      i       1      100      standard ANSYS file header (see binhed8 for
c                                   details of header contents)


c  ---      i       1      100      .MODE FILE HEADER
c
c                                    fun09,    nmrow,        0,    nmode,   numdof,  <  5
c                                     maxn,    wfmax,   lenbac,  nEnfGrp,    neqns,  < 10
c                                   lumpms,   extopt,   SvCode,      kan,   ldstep,  < 15
c                                   numitr,   expbeg,   expend,   nspect,   nSPdat,  < 20
c                                        0,   ptrFRQ, kPerturb,   ptrSHP,   ptrLOD,  < 25
c                                   ptrNAR,  ptrNARh,   ptrDMP,nrkeyPert,   nrigid,  < 30
c                                   ptrLPM,   ptrSP1,  ptrSHPh,  ptrLODh,        0,  < 35
c                                        0,  ptrDMPh,  ptrLPMh,  ptrSP1h, ptrIRHSl,  < 40
c                                 ptrIRHSh,        0,   ptrRES,  ptrRESh,Glblenbac,  < 45
c                                KeyStress,   ptrELD,  ptrELDh,   ptrGBk,  ptrGBkh,  < 50
c                                  modlstp,    nresi,   ptrEf1,  ptrEf1h,    sstif,  < 55
c                                  ptrFSTA,   ptrEf2,  ptrEf2h,   ptrEf3,  ptrEf3h,  < 60
c                                qrDampKey,cycMSUPkey,cycnmode,    ptrHI,  ptrKUNS,  < 65
c                                 ptrKUNSh, mrestart, LPrestls, LPrestss,   cpxmod,  < 70
c                                  keyLeft,    cpxlv,   ptrSCL, sparseLV, udfrqkey,  < 75
c                                ptrUDFRQl,ptrUDFRQh,        0,        0,        0,  < 80
c                                        0,        0,        0,        0,        0,  < 85
c                                        0,        0,        0,        0,        0,  < 90
c                                        0,        0,        0,        0,        0,  <100


c                                  each item in header is described below:

c                                   fun09  - unit number (mode file is 9)
c                                   nmrow  - number of rows/columns in matrices
c                                            (maxn*numdof)
c                                   nmode  - number of modes extracted
c                                   numdof - number of dof per node
c                                   maxn   - maximum node number (if extopt=3
c                                            or 4, the actual number of nodes is
c                                            referenced.)
c                                   wfmax  - maximum wavefront (does not apply
c                                            if extopt=3 or 4)
c                                   lenbac - number of nodes
c                                  nEnfGrp - numbre of enforced group
c                                   neqns  - number of equations on the .LN22
c                                            file
c                                   lumpms - lumped mass key
c                                            = 0 - default matrix type
c                                            = 1 - lumped
c                                            (does not apply if extopt=3 or 4)
c                                   extopt - mode extraction method
c                                            = 3 - unsymmetric Lanczos
c                                            = 4 - damped Lanczos
c                                            = 6 - block Lanczos
c                                            = 8 - SuperNode
c                                            = 9 - PCG Lanczos
c                                   SvCode - Solver assembly code path
c                                            = 1 Symbolic assembly
c                                   kan    - analysis type
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                          = 1 - buckling
c                                          = 2 - modal
c                                  ldstep - load step number - also number of load vectors
c                                  numitr - total number of cumulative
c                                           iterations done during analysis
c                                           (does not apply if extopt=3 or 4)
c                                  expbeg - beginning of the frequency range of
c                                           interest
c                                  expend - end of the frequency range of
c                                           interest
c                                  nspect - number of spectra; if -6, these are
c                                           the 6 default unit spectra
c                                  nSPdat - number of data items per spectrum
c                                  ptrFRQ - pointer to the frequencies
c                                kPerturb - Linear Perturbation key
c                                  ptrSHP - pointer to the mode shapes
c                                           (eigenvectors)
c                           ptrLOD, ptrLODh - pointer to the load vectors
c                           ptrNAR, ptrNARh - pointer to the nodal averaged result records
c                           ptrDMP, ptrDMPh - pointer to the modal damping matrix
c                         ptrKUNS, ptrKUNSh - pointer to the modal stiffness
c                                           matrix (unsymmetric part)
c                                 nrkeyPert - nrkey setting of base analysis (Linear Perturbation)
c                                    nrigid - number of rigid body modes
c                                    ptrLPM - pointer to the diagonal mass vector
c                                    ptrSP1 - pointer to the the spectrum data
c                                ptrIRHSl,h - pointer to imaginary part of RHS vector
c                             ptrRES,ptrRESh - pointer to residual vectors
c                                  Glblenbac - global number of nodes (for D-ANSYS)
c                             ptrGBk,ptrGBkh - pointer to global nodal equivalence table
c                                    modlstp - multiple load step key
c                                      nresi - number of residual vectors in file
c                                  KeyStress - key set if mode stresses on file
c                             ptrELD,ptrELDh - pointer to element records
c                             ptrEf1,ptrEf1h - pointer to enforced motion dof information
c                                      sstif - key denoting prestress effects are included,
c                                           this key is for internal usage only. SSTIF,on
c                                           or off is controlled by NLGEOM on or off now.
c                                    ptrFSTA - pointer to fstacm data
c                             ptrEf2,ptrEf2h - pointer to enforced motion modes
c                             ptrEf3,ptrEf3h - pointer to enforced motion force
c                                  qrDampKey - QR damped calcaulations key
c                                cycMSUPkey - mode file format is for subsequent cyclic MSUP
c                                           (only base results on file)
c                                   cycnmode - total number of cyclic modes extracted
c                                           (sum of all harmonic indices)
c                                      ptrHI - pointer to harmonic indices
c                                   mrestart - key for modal restart (=0 none, =1 modal restart)
c                                   LPrestls - restarted load step (from linear perturbation)
c                                   LPrestss - restarted substep (from linear perturbation)
c                                     cpxmod - key for complex frequencies/shapes (0=no 1=yes)
c                                    keyLeft - key for LMODE writing (0=no 1=yes)
c                                      cpxlv - key for RHS vector in complex form (0=no 1=yes)
c                                           = 0 no  (before version 17.0)
c                                           = 1 yes (since version 17.0) ; ptrIRHSl,h is 0
c                                    ptrSCL - pointer to cyclic mode scale factors
c                                           (if scaled to unity, modnrm on modopt)
c                                   sparseLV - key if load vectors have been sparsified
c                                          0 - position not used
c                                   udfrqkey - Key for format and writing of undamped frequency record
c                        ptrUDFRQl,ptrUDFRQh - pointer to undamped frequency record

c  ---      i       1     numdof    Degrees of freedom per node
c                                   DOF reference numbers are:
c         UX  = 1, UY  = 2, UZ  = 3, ROTX= 4, ROTY= 5, ROTZ= 6, AX  = 7, AY  = 8
c         AZ  = 9, VX  =10, VY  =11, VZ  =12, GFV1=13, GFV2=14, GFV3=15, WARP=16
c         CONC=17, HDSP=18, PRES=19, TEMP=20, VOLT=21, MAG =22, ENKE=23, ENDS=24
c         EMF =25, CURR=26, SP01=27, SP02=28, SP03=29, SP04=30, SP05=31, SP06=32
c                                    (curdof(i),i=1,numdof)

c  ---      i       1     lenbac    Nodal equivalence table
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                    This table equates the number used for
c                                    storage to the actual node number.
c                                     (Back(i),i=1,lenbac)

c  GBK      i       1    Glblenbac   Global nodal equivalence table
c                                    This table equates the number used for
c                                    storage to the actual global node number.
c                                     (GlbBack(i),i=1,Glblenbac)

c  FSTA    dp       1     30         fstacm.inc information (mass and MofI)

c  HI       i       1     cycnmode   Signed harmonic index for each extracted frequency.
c                                    Only present if cycMSUPkey=1.

c  FRQ     dp/cmp   1     nf         Frequencies (eigenvalues).
c                                    Frequencies are complex if cpxmod=1 or qrdampKey=1.
c                                    If frequencies are real, numbers stored are
c                                       the squares of the natural circular
c                                       frequencies (w**2, where w=radians/time).
c                                       You can obtain the natural frequencies, f
c                                       (in cycles/time), using the equation f=w/2pi
c                                    If frequencies are complex, numbers stored are
c                                       the natural frequencies (Hz)
c                                    (freq(i),i=1,nf)
c                                        nf = nmode+nresi
c                                      if cycMSUPkey=1 then
c                                        nf = cycnmode

c  UDFRQ     dp    1     nf         Undamped Eigenvalues for a QRDAMP Modal Analysis.
c                                    Record only exists when udfrqkey=1.
c                                    Numbers are stored as the squares of the natural
c                                    circular frequencies.
c                                    This record is directly written to the mode file and might
c                                    not be present in the .modesym file

c  SHP     dp/cmp   ns    nmrow      Mode shapes (eigenvectors). Mode shapes are
c                                    complex if cpxmod=1.  The order corresponds
c                                    to the nodal equivalence table
c                                     (psi(i,j),i=1,nmrow)
c                                        ns = nmode
c                                     if cycMSUPkey=1 then
c                                        ns = cycnmode

c  RES      dp    nresi   nmrow      Residual vectors

c  LOD     cmp   ldstep   nmrow      Load vectors in complex form (since version 17.0)
c                                     (f(i),i=1,nmrow)
c                                    Before version 17.0, records were (cpxlv=0):
c                                    LOD  dp ldstep nmrow Load vectors
c                                                          (f(i),i=1,nmrow)
c                                    IRHS dp ldstep nmrow Imaginary Load vectors
c                                                          (fimag(i),i=1,nmrow)

c  LPM      dp      1     nmrow      Lumped mass vector. This record is present
c                                    only if lumpms=1 and nmatrix=0. It is a
c                                    vector containing the mass at each node in
c                                    the system.
c                                     (mass(i),i=1,nmrow)

c  DMP      dp    nmrow   nmrow      Modal damping matrix. Each row of the matrix
c                                    matrix is stored as a record.
c                                     (ac(i,j),i=1,nmrow)

c  KUNS     dp    nmrow   nmrow      Modal unsymmetric stiffness matrix. Each row of the
c                                    matrix is stored as a record.
c                                     (aku(i,j),i=1,nmrow)

c  EF1     int      1    nEnfGrp    (groupID(i),i=1,nEnfGrp)
c          int      1    nEnfGrp    (grpdof(i),i=1,nEnfGrp)
c           dp  nEnfGrp  grpdof(i)  dofIndx(i,j) i=1,grpdof(j)
c                                    The above records contain information about each
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                  enforced motion group.

c  EF2      dp  nEnfGrp   nmrow     Enforced static modes

c  EF3      dp  nEnfGrp   nmrow     Enforced forced vector

c for each spectrum (|nspect| records):
c  SP1      dp      1   nmode+nresi Participation factors for this spectra
c  ---      dp      1   nmode+nresi Mode coefficients for this spectra
c  ---      dp      1   nmode+nresi Modal damping values
c  ---      dp      1    ndsvc*       (*) see svcom.inc
c  ---      int     1    nisvc*
c  ---      dp      1     20        misc. spectra data

c  ELD      int     1     15         nelm,  maskl,  nItems, ptrELM, ptrERS,
c                               ptrCERl,ptrCERh, ptrESLl,ptrESLh,    nRF,    (10)
c                                ptrRFl, ptrRFh, PrecKey,  maskh,      0,

c                                each item in header is described below:

c                                  nelm - number of elements
c                                maskl,h - output mask (OUTRES)
c                                 nItems - number of element records (7, VOL
c                                          not included)
c                                 ptrELM - pointer to element equivalence table
c                                 ptrERS - pointer to element record sizes
c                               ptrCERl,h - pointer to constant element records
c                               ptrESLl,h - pointer to element index
c                                     nRF - number of reaction forces
c                                ptrRFl,h - pointer to reaction forces
c                                 PrecKey - 0, double precision 1, single
c                     above pointers are relative to ptrELD

c  ---      int     1   2*nItems   Total size of each element record (LONGINT)

c  ELM      int     1     nelm     Element equivalence table
c                                   This table equates the order number used to
c                                   the actual element number

c  ERS      int   nItems  nelm     Sizes of the nItem element results sets for
c                                   each element

c  CER      int     1     5       ptrVOL, ptrEPT, ptrEUL,      0,      0
c                     above pointers are relative to ptrCER

c     constant element records (do not vary by mode):
c   VOL     dp      1  nelm*1       Element volume
c   EPT     dp      1  nelm*size    Element structural nodal temperatures
c   EUL     dp      1  nelm*size    Element Euler angles

c   ESL     int     1     10      ptrENS, ptrEEL, ptrEMS, ptrENF, ptrENG,
c                                 ptrENSh,ptrEELh,ptrEMSh,ptrENFh,ptrENGh
c                     above pointers are relative to ptrESL

c     non-constant element records (do vary by mode):
c   ENS     dp   nelm  nmode*size   Element nodal component stresses
c   EEL     dp   nelm  nmode*size   Element nodal component elastic strains
c   EMS     dp   nelm  nmode*size   Element summable miscellaneous data
c   ENF     dp   nelm  nmode*size   Element nodal forces
c   ENG     dp   nelm  nmode*3      Element energies

c     see fdresu.inc for more information on the element results

c  NAR      int     1      7         nnod, ptrNCTl, ptrNCTh, ptrNSTl, ptrNSTh,
c                                    ptrNELl, ptrNELh

c                                  each item in header is described below:

c                                    nnod - number of nodes
c                                ptrNCTl,h - pointer to node contributions
c                                ptrNSTl,h - pointer to NAR stress record
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                               ptrNELl,h - pointer to NAR elastic strain record
c                      above pointers are relative to ptrNAR

c     NAR records (do vary by mode):
c   NCT      i   nnod  2            Node Contributions
c   NST     dp   nnod  nmode*6      Nodal component stresses
c   NEL     dp   nnod  nmode*7      Nodal component elastic strains

c     see fdresu.inc for more information on the NAR records
```

1.6. Description of the Element Matrices File

This section explains the content of the element matrices file (`jobname.emat`).

1.6.1. Standard Mechanical APDL File Header

See The Standard Header for Mechanical APDL Binary Files (p. 4) for a description of this set. File number (Item 1) is 2.

1.6.2. EMAT File Format

```
*comdeck,fdemat

c *** Copyright ANSYS.  All Rights Reserved.
c *** ansys, inc.

c     **********  description of element matrix file  **********
c
      character*8  EMATNM
      parameter  (EMATNM='emat    ')

      integer         EMATHDLEN
      parameter      (EMATHDLEN=80)

      LONGINT         ematfpL, ematfp
      integer         ematbk, ematut, maxldset
      common /fdemat/ ematfpL, ematbk, ematut, maxldset
      equivalence  (ematfp,ematfpL)

c   ********** common variable descriptions ***********
co  ematfpL      file position on file emat
co  ematbk       block number for file emat
co  ematut       file unit for file emat

c   See fddesc for documentation of how binary files are stored.
c
c     **********  file format  **********

c        recid tells the identifier for this record.  Not all records will have
c             identifiers -- they are only indicated for those records whose
c             record pointers are stored in the second file header.

c        type tells what kind of information is stored in this record:
c             i - integer
c             dp - double precision
c             cmp - complex

c        nrec tells how many records of this description are found here

c        lrec tells how long the records are (how many items are stored)

c recid    type    nrec    lrec     contents
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c  ---      i       1      100      standard ANSYS file header (see binhed8 for
c                                   details of header contents)

c  ---      i       1       80      .EMAT FILE HEADER
c
c                                    fun02,    nume,   numdof,    lenu,  lenbac,
c                                     maxn, nlgeEMA,  sstEMAT,       0,   lumpm,
c                                    kygst,    kygm,     kycd,   kygss,   kygaf,
c                                    kygrf,       0,Glblenbac, ptrGBkl, ptrGBkh,
c                                  ptrElmh,       0,        0, ptrBITh, ptrEHDh,
c                                  ptrIDXh,   numCE,  maxLeng,  ptrCEl,  ptrCEh,
c                                   ptrDOF,  ptrBAC,  ptrELMl,       0,       0,
c                                  ptrBITl, ptrEHDl,  ptrIDXl, ptrendH, ptrendL,
c                                   nldstp,maxldset,  ptrLSIl, prtLSIh,       0,
c                                        0,       0,        0,       0,       0,
c                                        0,       0,        0,       0,       0,
c                                        0,       0,        0,       0,       0,
c                                        0,       0,        0,       0,       0,
c                                        0,       0,        0,       0,       0,
c                                        0,       0,        0,       0,       0,
c                                        0,       0,        0,       0,       0,

c                                 each item in header is described below:

c                                    fun02  - unit number (emat file is 2)
c                                    nume   - number of elements
c                                    numdof - number of dofs per node
c                                    lenu   - total DOFs of model
c                                    lenbac - number of nodes
c                                    maxn   - maximum node number
c                                    nlgeEMA  = 0 - nlgeom is OFF the time this Emat file is created
c                                               1 - nlgeom is ON the time this Emat file is created
c                                    sstEMAT  = 0 - sstif key is OFF the time this Emat file is created
c                                               1 - sstif key is ON the time this Emat file is created
c                                                   this key is for internal use only
c                                    lumpm  - lumped mass key
c                                             = 0 - default matrix type
c                                             = 1 - lumped
c                                    kygst  - global stiffness matrix calculate
c                                             key
c                                             = 0 - do not calculate
c                                             = 1 - calculate
c                                    kygm   - global mass matrix calculate key
c                                             = 0 - do not calculate
c                                             = 1 - calculate
c                                    kycd   - global damping matrix calculate key
c                                             = 0 - do not calculate
c                                             = 1 - calculate
c                                    kygss  - global stress stiffening matrix
c                                             calculate key
c                                             = 0 - do not calculate
c                                             = 1 - calculate
c                                    kygaf  - global applied force vector
c                                             calculate key
c                                             = 0 - do not calculate
c                                             = 1 - calculate
c                                    kygrf  - global restoring force vector
c                                             calculate key (Newton-Raphson only)
c                                             = 0 - do not calculate
c                                             = 1 - calculate
c                                    0      - position not used
c                                    Glblenbac - global global number of nodes (== lenbac unless using
c                                                Distributed Ansys)
c                                    ptrGBkl- low pointer to global nodal equivalence table
c                                    ptrGBkh- high pointer to global nodal equivalence table
c                                    ptrELMh- high pointer to element equivalence table
c                                    ptrBITh- high pointer to dof bits
c                                    ptrEHDh- high pointer to the start of the
c                                             element matrices
c                                    ptrIDXh- high pointer to element matrices
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                          index table
c                                  numCE  - number of internal CEs
c                                  maxLeng- maximum length of any internal CE
c                                  ptrCEl - low pointer to internal CE list
c                                  ptrCEh - high pointer to internal CE list
c                                  ptrDOF - pointer to degrees of freedom per
c                                           node used in model
c                                  ptrBAC - pointer to nodal equivalence table

c                                  ptrELMl- Low pointer to element equivalence
c                                           table
c                                  ptrBITl- Low pointer to dof bits
c                                  ptrEHDl- Low pointer to the start of the
c                                           element matrices
c                                  ptrIDXl- Low pointer to element matrices
c                                           index table

c                                  ptrendH- High pointer to end of file
c                                  ptrendL- Low  pointer to end of file
c
c                                  nldstp  - number element load vector set
c                                  maxldset- max eload vector set
c                                  ptrLSIl - location of Load Step Index
c                                  prtLSIh -

c       Note: the analysis type sets the global calculate keys.

c  ---     dp       1       20      Time information
c
c                                  timval, timinc, frqval, timbeg, timend,
c                                     0.0,    0.0,    0.0,    0.0,    0.0,
c                                     0.0,    0.0,    0.0,    0.0,    0.0,
c                                     0.0,    0.0,    0.0,    0.0,    0.0,

c                                 each item is described below:

c                                  timval - the current time
c                                  timinc - the time increment
c                                  frqval - the current frequency (from a
c                                           harmonic analysis)
c                                  timbeg - the start time for the analysis
c                                  timend - the end time for the analysis
c                                  0.0    - position not used
c                                  0.0    - position not used
c                                  0.0    - position not used
c                                  0.0    - position not used
c                                  0.0    - position not used
c                                  0.0    - position not used
c                                  0.0    - position not used
c                                  0.0    - position not used
c                                  0.0    - position not used
c                                  0.0    - position not used
c                                  0.0    - position not used
c                                  0.0    - position not used
c                                  0.0    - position not used
c                                  0.0    - position not used
c                                  0.0    - position not used

c  DOF      i       1    numdof     Degrees of freedom per node
c                                  DOF reference numbers are:
c         UX  = 1, UY  = 2, UZ  = 3, ROTX= 4, ROTY= 5, ROTZ= 6, AX  = 7, AY  = 8
c         AZ  = 9, VX  =10, VY  =11, VZ  =12, GFV1=13, GFV2=14, GFV3=15, WARP=16
c         CONC=17, HDSP=18, PRES=19, TEMP=20, VOLT=21, MAG =22, ENKE=23, ENDS=24
c         EMF =25, CURR=26, SP01=27, SP02=28, SP03=29, SP04=30, SP05=31, SP06=32
c                                   (curdof(i),i=1,numdof)

c  BAC      i       1    lenbac     Nodal equivalence table. This table equates
c                                  the number used for storage to the actual
c                                  node number
c                                   (Back(i),i=1,lenbac)
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c  ELM      i       1     nume      Element equivalence table. The ANSYS program
c                                  stores all element data in the numerical
c                                  order that the SOLUTION processor solves the
c                                  elements.  This table equates the order
c                                  number used to the actual element number
c                                   (Order(i),i=1,nume)

c  GBK      i       1  Glblenbac    Global nodal equivalence table.  This
c                                  table equates the number used for storage
c                                  to the actual node number.  Only written
c                                  by the master process in Distributed Ansys
c                                   (GlbBack(i),i=1,Glblenbac)

c  BIT      i       1     lenu      Bits set at a DOF table. This record
c                                  has bits for constraints, forces, etc.
c                                   (DofBits(i),i=1,lenu) (added at 10.0)

c  IDX      i       1     2*nume    Element index table. This record specifies
c                                  the file location for the beginning of the
c                                  data for each element.
c                                   (index(i),i=1,nume) Low part of pointer
c                                   (index(i),i=1,nume) High part of pointer


c  The records at the end of the file store element information and get written
c  as a set for each element(nume sets of these records will appear on the file
c  at this point) ptrEHD indicates the beginning of the element data.

c  If substructure matrices are written to the EMAT file, they are written in a
c  different format than is shown here. This alternate format is not documented
c  at this time, as it is likely to change in the future.


c  EHD      i       1      10       Element matrix header
c
c                                    stkey,   mkey,   dkey,  sskey, akey,
c                                    nrkey,   ikey,  kckey,      0, nmrow

c                                  each item in header is described below:

c                                    stkey  - stiffness matrix key
c                                             = 0 - matrix not present
c                                             = 1 - matrix present
c                                    mkey   - mass matrix key
c                                             = 0 - matirx not present
c                                             = 1 - matrix present
c                                    dkey   - damping matrix key
c                                             = 0 - matrix not present
c                                             = 1 - matrix present
c                                    sskey  - stress stiffening matrix key
c                                             = 0 - matrix not present
c                                             = 1 - matrix present
c                                    akey   - applied load vector key
c                                             = 0 - vector not used
c                                             = 1 - vector used
c                                    nrkey  - newton-raphson(restoring) load
c                                             vector key (for nonlinear analyses)
c                                             = 0 - vector not used
c                                             = 1 - vector used
c                                    ikey   - imaginary load vector key
c                                               (for complex analyses)
c                                             = 0 - vector not used
c                                             = 1 - vector used
c                                    kckey    = 0 or 1 or 2 position for internal use or not in use
c                                             = 3 complex number stiffness matrix key
c
c                                    0      - position not used
c                                    nmrow  - numbers/columns in matrices. If the
c                                             number is negative, the matrices
c                                             will be written in upper triangular
c                                             form.
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c  ---      i       1    nmrow      DOF index table. This record specifies the
c                                    DOF locations of this element matrix in
c                                    relation to the global matrix. The index is
c                                    calculated as (N-1)*NUMDOF+DOF, where N is
c                                    the position number of the node in the nodal
c                                    equivalence table and DOF is the DOF
c                                    reference number given above

c  ---     dp    varies  varies     Element matrices. This record is repeated
c                                    for each stiffness, mass, damping,stress stiffening
c                                    and complex stiffness matrice. If the matrix is
c                                    diagonal, the length of the records will be
c                                    nmrow.  If the matrix is unsymmetric, the
c                                    length of the records will be nmrow*nmrow.
c                                    If the matrix is symmetric, only the upper
c                                    triangular terms are written and the length
c                                    of the records will be (nmrow)*(nmrow+1)/2.

c  ---     dp       1    2*nmrow    Element force vectors. This record contains
c                                    both the applied force vector and the
c                                    (restoring or imaginary) load vector.
c
c
c      ************** Internal CE information ***********************
c      The following records repeat numCE times... one for each internal
c      CE created during solution... these are stored here for the
c      usage of a prestressed modal analysis such as the linear perturbation analysis
c
c  CE      i       3     numCE      First part is the CE number, the second part is
c                                    the number of terms in this internal CE, and
c                                    the third part is the external element number
c                                    of the element that created this internal CE
c
c  ---     i     nTerms  numCE      integer info (list of node*32 + dof)
c
c  ---    dp     nTerms  numCE      dp info (list of coefficients including constant term)
c
c
c   kygst        global stiffness matrix calculate key
c   kygm         global mass matrix calculate key
c   kygd         global damping matrix calculate key
c   kygss        global stress stiffening matrix calculate key
c   kygaf        global applied force matrix calculate key
c   kygrf        global restoring force matrix calculate key
c
c
c
c  Additional element records stored to support topo optimization
c  with multiple load steps. Currently only element load vectors are needed,
c  but keep the same format of the original element matrices/load vectors
c  for possible future need.
c
c  header(41) - header(44) are all 0 unless it's topo opitimization analysis
c
c  LSI      i       1     2*maxldset Load index table. This record specifies
c                                     the file location for the beginning of the
c                                     load steps after load step 2. In each load
c                                     steps, we have the following two records,
c                                     TPIDX and element force vectors
c
c  TPIDX    i       1     2*nume     Topo index table. This record specifies
c                                     the file location for the beginning of the
c                                     data for each element record, EHD.
c                                      (index(i),i=1,nume) Low part of pointer
c                                      (index(i),i=1,nume) High part of pointer
c                                     same as IDX above
c
c   EHD     i       1     10         Element matrix header
c                                     same as EHD above
c
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

```
c  ---      i       1     nmrow      DOF index table
c
c  ---     dp       1     2*nmrow    Element force vectors. This record contains
c                                    both the applied force vector and the
c                                    (restoring or imaginary) load vector.
```

1.7. Description of the Substructure Matrices File

This section explains the contents of the substructure matrices file (`jobname.sub`).

1.7.1. Standard Mechanical APDL File Header

See The Standard Header for Mechanical APDL Binary Files (p. 4) for a description of this set. File number (Item 1) is 8.

1.7.2. SUB File Format

```
*comdeck,fdsub
c
c *** Copyright ANSYS.  All Rights Reserved.
c *** ansys, inc

c     **********  description of substructure matrix file  **********
      character*8  SUBNM
      parameter  (SUBNM='sub    ')

      integer        SUBHDLEN
      parameter     (SUBHDLEN=80)

      LONGINT        subfpL, lenSubL
      integer        subbk, subut
      common /fdsub/ subfpL, lenSubL, subbk, subut

c  write:  matout
c  read:

c   ********** common variable descriptions ***********
co  subfpL      file position on file sub
co  subbk       block number for file sub
co  subut       file unit for file sub
co  lenSubL     length of sub file (saved for slvdta.F)

c   See fddesc for documentation of how binary files are stored.
c
c     **********  file format  **********

c     recid tells the identifier for this record.  Not all records will have
c          identifiers -- they are only indicated for those records whose
c          record pointers are stored in the second file header.

c     type tells what kind of information is stored in this record:
c          i - integer
c          dp - double precision
c          cmp - complex

c     nrec tells how many records of this description are found here

c     lrec tells how long the records are (how many items are stored)

c recid    type    nrec    lrec     contents
```

```
c  ---      i       1      100      standard ANSYS file header (see binhed8
c                                   for details of header contents)

c  HED      i       1       80      .SUB FILE HEADER  (FULL MATRICES)
c
c                                     8,  nmrow,  nmatrx,  nedge, numdof,
c                                   maxn,  wfmax,  lenbac,   nnod, kunsym,  10
c                                   kstf,  kmass,   kdamp,    kss,  nvect,
c                                 nWorkL,  lenU1,  sesort, lenlst,ptrLodL,  20
c                                 ntrans, ptrMtx,  ptrXFM, ptrHED,  name1,
c                                  name2,  ptrCG,       0,  name3,  name4,  30
c                                 ptrDOF, ptrDST,  ptrBAC, ptrTIT, ptrNOD,
c                                 ptrXYZ, ptrEDG,  ptrGDF, thsubs, ptrPOS,  40
c                                 ptrORG, stfmax, ptrLodH, nmodes, keydim,
c                               cmsMethod,  name5,   name6,  name7,  name8,  50
c                                 nvnodes,ptrCTXM,  nWorkH,      0,ptrTVAL,
c                                gyroDamp,kstress,nStartVN,ptrEndL,ptrEndH,  60
c                              ptrimsSEdat,ptrdmsSEdat,units,     0,      0,
c                                       0,      0,       0,      0,      0,  70
c                                       0,      0,       0,      0,      0,
c                                       0,      0,       0,      0      0   80


c  HED      i       1       80      .SUB FILE HEADER  (SPARSE MATRICES)
c
c                                     9,   nEqn, nmatrx,  ndege, numdof,
c                                   maxn,  wfmax, lenbac,   nnod, kunsym,  10
c                                   kstf,  kmass,  kdamp,       ,  nvect,
c                                 nTermL,       ,       , lenlst,ptrLodL,  20
c                                 ntrans,ptrMtxL, ptrXFM, ptrHED,  name1,
c                                  name2,  ptrCG,       ,  name3,  name4,  30
c                                 ptrDOF,       , ptrBAC, ptrTIT, ptrNOD,
c                                 ptrXYZ, ptrEdg, ptrGDF, thsubs,       ,  40
c                                       , stfmax,ptrLodH,       , keydim,
c                                       ,  name5,  name6,  name7,  name8,  50
c                                       ,ptrCTXM, nTermH,ptrMtxH,ptrColL,
c                                  ptrColH,ptrCofL,ptrCofH,ptrEndL,ptrEndH,  60
c                                       ,       ,       ,       ,       ,
c                                       ,       ,       ,       ,       ,
c                                       ,       ,       ,       ,       ,  70
c                                       ,       ,       ,       ,       ,
c                                       ,       ,       ,       ,          80

c                                  each item in header is described below:

c                                   fun08  - unit number (full sub file is 8)
c                                            (sparse substructure file is 9)
c                                   nmrow  - number of rows in matrices (also
c                                            number of dofs in substructure)
c                                   nmatrx - number of matrices on file
c                                   nedge  - number of edges for outline
c                                   numdof - number of dofs per node
c                                   maxn   - maximum node number of complete
c                                            model presently in database
c                                   wfmax  - maximum wavefront of substruct.
c                                            during generation pass
c                                   lenbac - number of nodes defining
c                                            substructure during the
c                                            generation pass
c                                   nnod   - number of unique nodes in the
c                                            substructure having DOFs, and
c                                            which define this substructure
c                                            during the use pass.  Also, the
c                                            number of nodes having master
c                                            DOFs.
c                                   kunsym - unsymmetric matrix key
c                                            = 0 - symmetric
c                                            = 1 - unsymmetric
c                                   kstf   - stiffness matrix present key
c                                            = 0 - matrix is not on file
c                                            = 1 - matrix is on file
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                kmass  - mass matrix present key
c                                         = 0 - matrix is not on file
c                                         = 1 - matrix is on file
c                                         =-1 - Lumped mass vestor (Sparse only)
c                                kdamp  - damping matrix present key
c                                         = 0 - matrix is not on file
c                                         = 1 - matrix is on file
c                                kss    - stress stiffening matrx present
c                                         = 0 - matrix is not on file
c                                         = 1 - matrix is on file
c                                nvect  - number of load vectors
c                                         (at least 1 is required)
c                                nWorkL,H  - BCS workspace length (only for
c                                            bacsub)
c                                nTermL,H  - Number of terms in sparse matrix
c                                lenU1  - length of intermediate transformation
c                                         vector
c                                sesort - DOF set sort key
c                                         = 0 - numbers are not sorted
c                                         = 1 - numbers are sorted in
c                                               ascending order
c                                lenlst - maximum length of DOF set for
c                                         this substructure (maxn*numdof)
c                                ptrLod - pointer to the start of the load
c                                         vectors (see also ptrLodh)
c                                ntrans - transformed key
c                                         = 0 - substructure has not been
c                                               transformed
c                                         > 0 - substructure copied
c                                               from another substructure,
c                                               via either SESSYM or SETRAN
c                                ptrMtxL,H - pointer to the start of the
c                                            substructure matrices (iDiagL for
c                                            sparse matrices)
c                                ptrXFM - pointer to the substructure
c                                         transformations
c                                ptrHED - pointer to the SUB file header
c                                name1  - first four characters of the
c                                         substructure file name, in
c                                         integer form
c                                name2  - second four characters of the
c                                         substructure file name, in
c                                         integer form
c                                name3  - third four characters of the
c                                         substructure file name, in
c                                         integer form
c                                name4  - fourth four characters of the
c                                         substructure file name, in
c                                         integer form
c                                ptrDOF - pointer to the DOF/node list
c                                ptrDST - pointer to the local DOF set
c                                ptrBAC - pointer to the nodes comprising
c                                         the substructure
c                                ptrTIT - pointer to the title
c                                ptrNOD - pointer to the unique nodes
c                                         defining the substructure
c                                ptrXYZ - pointer to the coordinates of the
c                                         unique nodes
c                                ptrEDG - pointer to the substructure edges
c                                ptrGDF - pointer to the global DOF set
c                                ptrCG  - pointer to the element mass information
c                                thsubs - element type key
c                                         = 0 - structural
c                                         = 1 - 1st order non-structural
c                                               (generally from thermal)
c                                         = 2 - 2nd order non-structural
c                                ptrPOS - pointer to the sorted substructure
c                                         DOF set to the original
c                                ptrORG - pointer to the DOF set of the model
c                                         during the generation pass
c                                stfmax - maximum diagonal stiffness term
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                       (packed into an integer)
c                               ptrLodh- High 32 bits of 64 bit pointer
c                               nmodes - number of modes used to generate
c                                       CMS s.e.
c                               keydim - dimensionality key
c                                       = 1 - axisymmetric
c                                       = 2 - 2-D
c                                       = 3 - 3-D
c                             cmsMethod - component mode synthesis method
c                                name5  - fifth four characters of the
c                                       substructure file name, in integer
c                                       form
c                                name6  - sixth four characters of the
c                                       substructure file name, in integer
c                                       form
c                                name7  - seventh four characters of the
c                                       substructure file name, in integer
c                                       form
c                                name8  - eighth four characters of the
c                                       substructure file name, in integer
c                                       form
c                              nvnodes - number of virtual nodes that contain
c                                       the modal coordinates
c                             gyroDamp - gyroscopic damping matrix key
c                                       = 0 - damping matrix is
c                                             not skew-symmetric if present
c                                       = 1 - damping matrix is
c                                             skew-symmetric, due to
c                                             gyroscopic effect
c                               kStress - = 1 if modal element results are
c                                       on the .cms file
c                              nStartVN - starting number of the virtual nodes
c                                       if defined with CMSOPT command
c                               ptrCTXM - coordinate transformation
c                             ptrColL,H - pointer to the iCol sparse matrix
c                                         array
c                             ptrCofL,H - pointer to the of the
c                                         sparse matrix Sk(1:nTerm),
c                                         Sm(1:nTermL),Sc(1:nTermL),
c                                         Ss(1:nTermL)  Each matrix is a
c                                         single large record
c                             ptrEndL,H - Next location after end of file
c                            ptrimsSEdat - pointer to integer record for stage data
c                            ptrdmsSEdat - pointer to double precision records for stage data
c                                units  - unit system used
c                                       =-1 - no /UNITS specification
c                                       = 0 - user defined units
c                                       = 1 - SI
c                                       = 2 - CSG
c                                       = 3 - U.S. Customary, using feet
c                                       = 4 - U.S. Customary, using inches
c                                       = 5 - MKS
c                                       = 6 - MPA
c                                       = 7 - uMKS

c                               note: name1/2/3/4/5/6/7/8 are the
c                                     inexc4 representation of the
c                                     32 character filename.
c                                     name1/2/5/6/7/8 will be "0"
c                                     for pre rev 5.2 files - cwa

c   Note: If ntrans > 0, records from position ptrDOF to ptrGDF will be
c         identical to the data for the copied substructure.


c  XFM      dp      1      125   Substructure transformations (5*25 double
c                                precisions). This record has meaning only
c                                if ntrans > 0. You can define up to five
c                                levels of transformations, with 25 variables
c                                in each level.  Up to the first seven
c                                variables are used as follows:
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

```
c
c                                If the substructure was transferred (via the
c                                 SETRAN command):
c                                1st variable - 1.0
c                                2nd variable - nodal increment
c                                3rd variable - reference number of
c                                 coordinate system where substructure will
c                                 be transferred
c                                4th variable - reference number of
c                                 coordinate system where substructure is
c                                 presently defined
c                                5th variable - x coordinate increment
c                                6th variable - y coordinate increment
c                                7th variable - z coordinate increment

c                                If the substructure used symmetry (via the
c                                 SESYMM command):
c                                1st variable - 2.0
c                                2nd variable - nodal increment
c                                3rd variable - number of coordinate
c                                 component to be used in operation
c                                 = 1 - x coordinate
c                                 = 2 - y coordinate
c                                 = 3 - z coordinate
c                                4th variable - reference number of
c                                 coordinate system to be used for symmetry
c                                 operation
c  CTXM     dp      1      250   Substructure transformations

c  DOF      i       1     numdof    Degrees of freedom per node  (Global)
c                                     (curdof(i),i=1,numdof)
c                                   DOF reference numbers are:
c    UX  = 1, UY  = 2, UZ  = 3, ROTX= 4, ROTY= 5, ROTZ= 6, AX  = 7, AY  = 8
c    AZ  = 9, VX  =10, VY  =11, VZ  =12, GFV1=13, GFV2=14, GFV3=15, WARP=16
c    CONC=17, HDSP=18, PRES=19, TEMP=20, VOLT=21, MAG =22, ENKE=23, ENDS=24
c    EMF =25, CURR=26, SP01=27, SP02=28, SP03=29, SP04=30, SP05=31, SP06=32

c  DST      i       1     nmrow  This record contains degrees of freedom for
c                                this substructure of the unique nodes, as
c                                used with this substructure, in ascending
c                                order. This index is calculated as
c                                (N-1)*numdof+DOF, where N is the node number
c                                and DOF is the DOF reference number given
c                                above
c                                 (lsort(i),i=1,nmrow)

c  POS      i       1     nmrow  This record stores the positions of the
c                                local DOF set in relation to the generated
c                                DOF set. (lposit(i),i=1,nmrow)

c  ORG      i       1     nmrow  DOF set of the model as defined during the
c                                generation pass. This index is calculated as
c                                (N-1)*NUMDOF+DOF, where N is the position
c                                number of the node in the nodal equivalence
c                                table and DOF is the DOF reference number
c                                given above
c                                 (lorig(i),i=1,nmrow)

c  BAC      i       1     lenbac This group describes nodes that defined the
c                                substructure during the generation pass of
c                                the analysis. Nodal data is stored in arrays
c                                equal to the number of used or referenced
c                                nodes. This table equates the number used
c                                for storage to the actual node number.
c                                 (Back(i),i=1,lenbac)

c  TIT      i       1       20   Substructure title (converted to integers -
c                                see inexc4)

c  NOD      i       1      nnod  This record describes unique nodes defining
c                                the substructure for the use pass of the
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                  analysis.  These are also the nodes having
c                                  master degrees of freedom.
c                                   (node(i),i=1,nnod)

c  XYZ     dp     nnod      6     This record describes the coordinates of a
c                                  unique node, in the order X, Y, Z, THXY,
c                                  THYZ, and THZX.  Nodal order corresponds to
c                                  that of the node list given above
c                                   (xyzang(j,i),j=1,6)

c  EDG     dp     nedge     6     This record contains beginning and ending
c                                  locations (X1,Y1,Z1,X2,Y2,Z2 coordinates) of
c                                  a straight line comprising an edge of the
c                                  substructure.

c  GDF    LONG      1     nmrow  This record describes global degrees of
c                                  freedom of the unique nodes in ascending
c                                  order, as used during the analysis use pass.
c                                  This index is calculated as (N-1)*32+DOF,
c                                  where N is the node number and DOF is the
c                                  DOF reference number given above
c                                   (l(i),i=1,nmrow) (sorted)
c                                  (Made LONGINT in rev 14.0)

c  CG      dp       1      37     Totmass,CGx,CGy,CGz,mm11,mm22,mm33,mm12,mm23,
c                                  mm13. And three precise mass matrices: MASSTR,
c                                  MASSROSP, and MASSTRRO. Each has 3*3 doubles.

c  TVAL    dp       1     1000    current time value corresponds to each load step
c                                  (substructuring analysis only)

c  IMSSE    i       1      17     integer record for stage data (see multiStage.inc):
c                                  msNsector,msHindex,msNumoff,
c                                  msCECYC(i)(i=1,2),msNodElm(i)(i=1,12)

c  DMSSE   dp       1       6     double precision record for stage data (see multiStage.inc):
c                                  msBox(i)(i=1,6)

c The substructure matrices are written at this position in the file.  One row
c  of each matrix is written to the file at a time. i.e. the first row of each
c  matrix is written, then the second row of each matrix, etc. this pattern
c  continues until all nmrow rows of each matrix have been written to the file.

c  MAT      dp      1     nmrow   Row of the stiffness matrix, if nmatrx > 0.
c                                   (ak(i,j),i=1,nmrow)
c  ---      dp      1     nmrow   Row of the mass matrix, if nmatrx > 1.
c                                   (am(i,j),i=1,nmrow)
c  ---      dp      1     nmrow   Row of the damping matrix, if nmatrx > 2.
c                                   (ac(i,j),i=1,nmrow)
c  ---      dp      1     nmrow   Row of the stress stiffening matrix, if
c                                  nmatrx > 3.
c                                   (gs(i,j),i=1,nmrow)

c  LOD      dp   nvect    nmrow   This record contains the load vectors.
c                                   (f(i),i=1,nmrow)
```

1.8. Description of the Component Mode Synthesis Matrices (CMS) File

This section explains the contents of the CMS matrices file (`jobname.cms`).

1.8.1. Standard Mechanical APDL File Header

See The Standard Header for Mechanical APDL Binary Files (p. 4) for a description of this set. File number (Item 1) is 8.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1.8.2. CMS File Format

```
*comdeck,fdcms

c *** Copyright ANSYS.  All Rights Reserved.
c *** ansys, inc.

c     **********  description of CMS (component mode synthesis) transformation file  **********

      character*8  CMSNM
      parameter   (CMSNM='cms     ')

      LONGINT         cmsfpL, cmsfp
      integer         cmsbk, cmsut

      common /fdcms/  cmsfpL, cmsbk, cmsut
      equivalence (cmsfp,cmsfpL)

c   ********** common variable descriptions ***********
co  cmsfp       file position on file cms
co  cmsbk       block number for file cms
co  cmsut       file unit for file cms


c   See fddesc for documentation of how binary files are stored.
c
c     **********  file format  **********

c        recid tells the identifier for this record.  Not all records will have
c             identifiers -- they are only indicated for those records whose
c             record pointers are stored in the second file header.

c        type tells what kind of information is stored in this record:
c             i - integer
c             dp - double precision
c             cmp - complex

c        nrec tells how many records of this description are found here

c        lrec tells how long the records are (how many items are stored)

c recid    type    nrec    lrec     contents

c  ---      i       1      100      standard ANSYS file header (see binhed8 for
c                                   details of header contents)

c  ---      i       1       40      .CMS FILE HEADER
c
c                                  fun45,    neqn,   nirfm,   nnorm,   ncstm,
c                                  nrsdm, cmsMeth, kStress,  lenbac,  numdof,
c                                cmsMixF,       0,       0,       0,       0,
c                                     0,       0,       0,       0,       0,
c                                     0,       0,       0,       0,       0,
c                                     0,       0,       0, ptrNARl, ptrNARh,
c                                ptrIRFl, ptrNORl, ptrCSTl, ptrRSDl, ptrIRFh,
c                                ptrNORh, ptrCSTh, ptrRSDh, ptrELDl, ptrELDh

c                                  each item in header is described below:
c                                   fun45  - unit number
c                                    neqn  - number of equations (DOF)
c                                   nirfm  - number of inertia relief modes
c                                   nnorm  - number of normal modes
c                                   ncstm  - number of constraint modes
c                                          > 0 available in file
c                                          < 0 not available in file
c                                   nrsdm  - number of residual modes
c                                 cmsMeth  - CMS method key
c                                              0 = fixed interface method
c                                              1 = free interface method
```

```
c                                              3 = residual-flexible free interface method
c                                              4 = user defined method
c                                  cmsMixF  - flags mixed interface method derived from
c                                              1 = free interface method
c                                              3 = Residual-flexible free interface method
c                                              0 = default
c                                   kStress - key if modal element results are on file
c                             ptrNARl,ptrNARh  - 64 bit pointer to nodal averaged result records
c                             ptrIRFl,ptrIRFh  - 64 bit pointer to inertia relief modes
c                             ptrNORl,ptrNORh  - 64 bit pointer to normal modes
c                             ptrCSTl,ptrCSTh  - 64 bit pointer to constraint modes
c                             ptrRSDl,ptrRSDh  - 64 bit pointer to residual modes
c                             ptrELDl,ptrELDh  - 64 bit pointer to element records
c                                     0      - position not used

c  ---       i       1    neqn      SOLVER-to-ANSYS mapping vector (lSOLVtoANS(i), i=1,neqn)

c                                     Note: When using the residual-flexible free interface method
c                                           the modes are written in ANSYS internal equation ordering
c                                           and the SOLVER-to-ANSYS mapping is not available.  For
c                                           other methods the modes are written in solver equation
c                                           ordering and the SOLVER-to-ANSYS mapping is available.

c  ---      i        1    lenbac    Nodal equivalence table. This table equates the number used
c                                     for storage to the actual node number.
c                                     (Back(i),i=1,lenbac)

c  NOR      dp   nnorm    neqn      Normal Modes
c
c  IRF      dp   nirfm    neqn      Inertia Relief Modes
c
c  CST      dp   ncstm    neqn      Constraint Modes
c
c  RSD      dp   nrsdm    neqn      Residual modes

c modal element results if kStress = 1:
c  ELD      int     1     15        nelm,   maskl,  nItems,  ptrELM, ptrERS,
c                                  ptrCER, ptrCERh,  ptrESL, ptrESLh,    nRF,    (10)
c                                   ptrFR,  ptrRFh, PrecKey,   maskh,      0

c                                   each item in header is described below:

c                                     nelm - number of elements
c                                   maskl,h - output mask (OUTRES)
c                                    nItems - number of element records (7, VOL
c                                             not included)
c                                   ptrELM - pointer to element equivalence table
c                                   ptrERS - pointer to element record sizes
c                                  ptrCER,h - pointer to constant element records
c                                  ptrESL,h - pointer to element index
c                                       nRF - number of reaction forces
c                                   ptrRF,h - pointer to reaction forces
c                                   PrecKey - 0, double precision 1, single
c                        above pointers are relative to ptrELD

c  ---      int     1   2*nItems   Total size of each element record (LONGINT)

c  ELM      int     1     nelm     Element equivalence table
c                                   This table equates the order number used to
c                                   the actual element number

c  ERS      int   nItems  nelm     Sizes of the nItem element results sets for
c                                   each element

c  CER      int     1     5       ptrVOL, ptrEPT, ptrEUL,      0,      0
c                        above pointers are relative to ptrCER

c     constant element records (do not vary by mode):
c   VOL     dp      1  nelm*1       Element volume
c   EPT     dp      1  nelm*size    Element structural nodal temperatures
c   EUL     dp      1  nelm*size    Element Euler angles
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

```
c   ESL     int     1     10      ptrENS, ptrEEL, ptrEMS, ptrENF, ptrENG,
c                                 ptrENSh,ptrEELh,ptrEMSh,ptrENFh,ptrENGh
c                     above pointers are relative to ptrESL

c     non-constant element records (do vary by mode). Modes order is:
c     --- nmode = [NOR CST (BCLV)]       for fix interface method
c     --- nmode = [NOR (IRF) CST (BCLV)] for free interface method
c     --- nmode = [NOR RSD]              for RFFB method
c     --- () = if any
c     --- BCLV = static correction vectors stored in .BCLV file
c
c   ENS     dp   nelm  nmode*size   Element nodal component stresses
c   EEL     dp   nelm  nmode*size   Element nodal component elastic strains
c   EMS     dp   nelm  nmode*size   Element summable miscellaneous data
c   ENF     dp   nelm  nmode*size   Element nodal forces
c   ENG     dp   nelm  nmode*3      Element energies

c     see fdresu.inc for more information on the element results

c  NAR      int     1      7         nnod, ptrNCTl, ptrNCTh, ptrNSTl, ptrNSTh,
c                                    ptrNELl, ptrNELh

c                                  each item in header is described below:

c                                    nnod - number of nodes
c                                ptrNCTl,h - pointer to Node Contributions
c                                ptrNSTl,h - pointer to NAR stress record
c                                ptrNELl,h - pointer to NAR elastic strain record
c                     above pointers are relative to ptrNAR

c     NAR records (do vary by mode):
c   NCT      i   nnod  2            Node Contributions
c   NST     dp   nnod  nmode*6      Nodal component stresses
c   NEL     dp   nnod  nmode*7      Nodal component elastic strains

c     see fdresu.inc for more information on the NAR records
```

1.8.3. TCMS File Format

```
*comdeck,fdtcms

c *** Copyright ANSYS.  All Rights Reserved.
c *** ansys, inc.

c     **********  description of CMS (component mode synthesis) transformation file  **********

      character*8  TCMSNM
      parameter   (TCMSNM='tcms     ')

      LONGINT         tcmsfpL, tcmsfp
      integer         tcmsbk, tcmsut

      common /fdtcms/ tcmsfpL, tcmsbk, tcmsut
      equivalence (tcmsfp,tcmsfpL)

c   ********** common variable descriptions ***********
co  tcmsfp       file position on file tcms
co  tcmsbk       block number for file tcms
co  tcmsut       file unit for file tcms


c   See fddesc for documentation of how binary files are stored.
c
c     **********  file format  **********

c        recid tells the identifier for this record.  Not all records will have
c             identifiers -- they are only indicated for those records whose
```

```
c             record pointers are stored in the second file header.

c        type tells what kind of information is stored in this record:
c             i - integer
c             dp - double precision
c             cmp - complex

c        nrec tells how many records of this description are found here

c        lrec tells how long the records are (how many items are stored)

c recid    type    nrec    lrec     contents

c  ---      i       1      100      standard ANSYS file header (see binhed8 for
c                                   details of header contents)

c  ---      i       1       40      .TCMS FILE HEADER
c
c                                  fun48,  nNodes,    neqn,  numdof,  nirfm,
c                                  nnorm,   ncstm,       0,       0,       0,
c                                ptrNORl, ptrCSTl, ptrIRFl,       0,       0,
c                                      0,       0,       0,       0,       0,
c                                ptrNORh, ptrCSTh, ptrIRFh,       0,       0,
c                                      0,       0,       0,       0,       0,
c                                      0,       0,       0,       0,       0,
c                                      0,       0,       0,       0,       0,
c                                      0,       0,       0,       0,       0

c                                  each item in header is described below:
c                                   fun48  - unit number
c                                  nNodes  - number of nodes in file
c                                    neqn  - number of equations (nNodes*numdof)
c                                  numdof  - number of dofs per node
c                                   nirfm  - number of inertia relief modes
c                                   nnorm  - number of normal modes
c                                   ncstm  - number of constraint modes
c                           ptrIRFl,ptrIRFh  - 64 bit pointer to inertia relief modes
c                           ptrNORl,ptrNORh  - 64 bit pointer to normal modes
c                           ptrCSTl,ptrCSTh  - 64 bit pointer to constraint modes
c                                    0       - position not used

c   ---      i       1  nNodes      Nodal equivalence table. This table equates
c                                   the number used for storage to the actual
c                                   node number

c   NOR     dp   nnorm    neqn      Normal Modes

c   IRF     dp   nirfm    neqn      Inertia Relief Modes

c   CST     dp   ncstm    neqn      Constraint Modes
```

1.9. Description of the Full Stiffness-Mass File

This section explains the contents of the full file (`jobname.full`).

1.9.1. Standard Mechanical APDL File Header

See The Standard Header for Mechanical APDL Binary Files (p. 4) for a description of this set. File number (Item 1) is 4.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1.9.2. FULL File Format

```
*comdeck,fdfull

c *** Copyright ANSYS.  All Rights Reserved.
c *** ansys, inc.

c     **********  description of full stiffness-mass file  **********

      character*8  FULLNM
      parameter  (FULLNM='full    ')

c     *** NOTE: if this variable is changed in the future it should be
c     ***       updated in spdefines.h also for symbolic assembly (jrb)
      integer         FULLHDLEN
      parameter      (FULLHDLEN=160)

      LONGINT         fullfpL, fullfp
      integer         fullbk,  fullut, wrLdstep, wrSbstep, wrEqiter,
     x                wrOption
      common /fdfull/ fullfpL, fullbk, fullut,
     x                wrLdstep,wrSbstep,wrEqiter,wrOption
      equivalence  (fullfp,fullfpL)

c   ********** common variable descriptions ***********
co  fullfpL      file position on file full
co  fullbk       block number for file full
co  fullut       file unit for file full

c     **********  file format (except for extopt=3,4) **********

c   See fddesc for documentation of how binary files are stored.

c     **********  file format  **********

c        recid tells the identifier for this record.  Not all records will have
c             identifiers -- they are only indicated for those records whose
c             record pointers are stored in the second file header.

c        type tells what kind of information is stored in this record:
c             i - integer
c             dp - double precision
c             cmp - complex

c        nrec tells how many records of this description are found here

c        lrec tells how long the records are (how many items are stored)

c recid    type    nrec    lrec     contents

c  ---      i       1      100      standard ANSYS file header (see binhed8 for
c                                   details of header contents)

c  ---      i       1      160      .FULL FILE HEADER
c
c                                  fun04,     neqn,    nmrow,   nmatrx,      kan,
c                                  wfmax,   lenbac,   numdof,  ntermKl,  ntermKh,   (10)
c                                  lumpm,    nmrow,   ntermK,   keyuns,   extopt,
c                                  keyse,   sclstf,   nxrows,  ptrSTFl,  ptrSTFh,   (20)
c                                ncefull,  ntermMh,  ptrENDl,  ptrENDh, ptrIRHSl,
c                               ptrIRHSh,  ptrMASl,  ptrMASh,  ptrDMPl,  ptrDMPh,   (30)
c                                 ptrCEl,   ptrCEh,   nNodes,  ntermMl,  ntermDl,
c                                ptrDOFl,  ptrDOFh,  ptrRHSl,  ptrRHSh,  ntermDh,   (40)
c                                ngMaxNZ, ptrNGPHl, ptrNGPHh, minKdiag, maxKdiag,
c                               minMdiag, maxMdiag, minDdiag, maxDdiag,  ngTerml,   (50)
c                                ngTermh, ngTermCl, ngTermCh,ptrDIAGKl,ptrDIAGKh,
c                              ptrDIAGMl,ptrDIAGMh,ptrDIAGCl,ptrDIAGCh, ptrSCLKl,   (60)
c                               ptrSCLKh,  Glbneqn,  distKey, ngTermFl, ngTermFh,
c                              GlbnNodes, GlbnVars, GlbfAcCE,  lcAcLen,   GlbfCE,   (70)
```

```
c                                ptrGmtl,  ptrGmth,nceGprime,numA12A11,GnVirtBCs,
c                                ntermGl,  ntermGh,ptrDensel,ptrDenseh, nVirtBCs,   (80)
c                             ptrVrtBCl,ptrVrtBCh,  ptrMRKl,  ptrMRKh, ptrKclxl,
c                              ptrKclxh, ntermKCl, ntermKCh,minKCdiag,maxKCdiag,   (90)
c                                 ngChg,   ptrBCl,   ptrBCh, ptrPHYSl, ptrPHYSh,
c                                predKey,fullDistF, ptrGVBCl, ptrGVBCh,    nzRow,  (100)
c                            localNonlKey, nMastDOF,  ptrMDFl,  ptrMDFh, GlbnMast,
c                                ptrGMDFl, ptrGMDFh,  cmsMeth,  cmsMixF,   hrmopt,  (110)
c                                 ActFlag,InActNKey,IANSndCnt,IANRcvCnt,Glblenbac,
c                                 ptrActl,  ptrActh,ext_nNods, ext_neqn,ext_nmrow,  (120)
c                              MtxEqnFlag,   krysub,  ext_nce, ptrExtNl, ptrExtNh,
c                              ExtSndCnt,ExtRcvCnt,        0,        0,        0,  (130)
c                                     0,        0,        0,        0,        0,
c                                     0,        0,        0,        0,        0,  (140)
c                                     0,        0,        0,        0,        0,
c                                     0,        0,        0,        0,        0,  (150)
c                                     0,        0,        0,        0,        0,
c                                     0,        0,        0,        0,        0   (160)


c                                  each item in header is described below:

c                                   fun04  - negative of the unit number (-4)
c                                            NOTE: if fun04 is > 0 it means that
c                                                  frontal assembly was used (which
c                                                  is longer documented here)
c                                   neqn   - number of equations on file
c                                   nmrow  - number of active DOF (neqn-BC)
c                                   nmatrx - number of matrices on file
c                                   kan    - analysis type
c                                   wfmax  - maximum row size
c                                   lenbac - number of nodes in ANSYS space (this can be different from nNodes wh
c                                   numdof - number of dofs per node
c                                   ntermKl,ntermKh - number of terms in Stiffness
c                                                     matrix
c                                   lumpm  - lumped mass key
c                                            = 0 - default matrix type
c                                            = 1 - lumped
c                                   ntermK - pre-8.1 this is the number of terms
c                                            in Stiffness matrix (otherwise this
c                                            value must be 0 and ntermKl,ntermKh
c                                            must be used)
c                                   keyuns - unsymmetric key
c                                            = 0 - no unsymmetric matrices on
c                                                  file
c                                            = 1 - there is at least one
c                                                  unsymmetric matrix on file
c                                   extopt - mode extraction method
c                                            = 0 - reduced
c                                            = 1 - lumped
c                                            = 3 - unsymmetric Lanczos
c                                            = 4 - damped Lanczos
c                                            = 6 - block Lanczos
c                                            = 7 - QRdamped
c                                            = 8 - SuperNode
c                                            = 9 - PCG Lanczos
c                                   keyse  - superelement key; set if at least
c                                           one superelement
c                                   sclstf - maximum absolute stiffness matrix term
c                                   nxrows - the maximum rank for this solution
c                                   ptrSTFl,h  - pointer to Stiffness matrix
c                                   ncefull    - number of CE+CP equations
c                                   ptrENDl    - low part of 64 bit end of file ptr
c                                   ptrENDh    - high part of 64 bit end of file ptr
c                                   ptrIRHSl,h - pointer to imaginary RHS (F)
c                                   ptrMASl,h  - pointer to Mass matrix
c                                   ptrDMPl,h  - pointer to Damping matrix
c                                   ptrCEl,h   - pointer to Gt and g matrices
c                                   nNodes     - number of nodes considered by assembly (nNodes can be different
c                                   ntermMl,h  - number of terms in Mass matrix
c                                   ntermDl,h  - number of terms in Damping matrix
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                  ptrDOFl,h  - pointer to DOF info
c                                  ptrRHSl,h  - pointer to RHS (F)
c                                  ngMaxNZ    - maximum number of nodes per nodal
c                                               block in nodal graph structure
c                                  ptrNGPHl,h - pointer to vectors needed for
c                                               nodal graph structure
c                                  minKdiag - minimum absolute stiffness matrix
c                                             diagonal term
c                                  maxKdiag - maximum absolute stiffness matrix
c                                             diagonal term
c                                  minMdiag - minimum absolute mass matrix
c                                             diagonal term
c                                  maxMdiag - maximum absolute mass matrix
c                                             diagonal term
c                                  minDdiag - minimum absolute damping matrix
c                                             diagonal term
c                                  maxDdiag - maximum absolute damping matrix
c                                             diagonal term
c                                  ngTerml,h  - total number of nonzeroes in nodal graph
c                                               (expanded graph based value, no BC applied)
c                                  ngTermCl,h - total number of nonzeroes in nodal graph
c                                               (compressed graph based value)
c                                  ptrDIAGKl,h - pointer to stiffness matrix DIAGONAL vector
c                                                (NOTE: this is a copy of the diagonal
c                                                       values stored in the full matrix)
c                                  ptrDIAGMl,h - pointer to mass matrix DIAGONAL vector
c                                                (NOTE: this is a copy of the diagonal
c                                                       values stored in the full matrix)
c                                  ptrDIAGCl,h - pointer to damping matrix DIAGONAL vector
c                                                (NOTE: this is a copy of the diagonal
c                                                       values stored in the full matrix)
c                                  ptrSCLKl,h  - pointer to stiffness matrix diagonal scaling
c                                                vector (may contain all 1.0's when not scaling)
c                                  Glbneqn  - global number of active DOF (this will match nmrow
c                                             unless we are writing distributed "local" FULL files
c                                             in Distributed ANSYS
c                                  distKey  - key denoting whether the FULL file is a single,
c                                             global FULL file (0) or multiple, local FULL file (1)
c                                  ngTermFl,h - total number of nonzeroes in nodal graph
c                                               as passed to the solver (after BC applied)
c                                  GlbnNodes - global number of nodes considered by assembly
c                                  GlbnVars  - global number of equations (will match neqn
c                                              unless we are writing distributed "local" FULL files
c                                              in Distributed ANSYS
c                                  GlbfAcCE  - total number of across CE
c                                  lcAcLen   - number of acrossCE where depedents are in my domain
c                                  GlbfCE    - total number of all the CE
c                                  ptrGmtl,h - pointer to G prime matrix for local nonlinearity
c                                  nceGprime - number of CE (or equations) in G prime local nonlinearity
c                                  numA12A11 - number of equations in local nonlinear matrix: excluding
c                                              equations from G prime
c                                  GnVirtBCs - global number of virtual constraints
c                                  ntermGl,ntermGh - total number of terms in total local nonlinear
c                                                    matrix including A12,A11 and G: total sum
c                                  ptrDensel,ptrDenseh - dense matrix information in local nonlinear
c                                                        matrix
c                                  nVirtBCs  - number of virtual constraints
c                                  ptrVrtBCl,ptrVrtBCh - pointer to the virtual constraint DOF data
c                                  ptrMRKl,h - pointer to the DOF marker array
c                                  0         - position not used
c                                  ptrKclxl,h - pointer to K complex (the 4th matrix) matrix
c                                               full case is: K, M, C, Kcplx: existing at same time
c                                  ntermKCl,h - number of terms in Complex Stiffness matrix
c                                  minKCdiag - minimum absolute complex stiffness matrix
c                                              diagonal term
c                                  maxKCdiag - maximum absolute complex stiffness matrix
c                                              diagonal term
c                                  ngChg     - key denoting whether or not the nodal graph written
c                                              to this FULL file differs from the previous FULL file
c                                  ptrBCl,h  - pointer to boundary condition data
c                                  ptrPHYSl,h- pointer to the physics marker array
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                   predKey   - nonlinear analysis predictor key
c                                   fullDistF - indicate the global profiling of distributed full file
c                                   ptrGVBCl,h- pointer to the global virtual constraint DOF data
c                                   nzRow     - number of non-zero rows of the matrics for the full file under
c                                               global full file, -1 means not applicable
c                                   localNonlKey - local nonlinearity speedup key
c                                   nMastDOF  - number of master DOF for substructuring
c                                   ptrMDFl,h - pointer to the master DOF data for substructuring
c                                   GlbnMast  - global number of master DOF for substructuring
c                                   ptrGMDFl,h- pointer to the global master DOF data for substructuring
c                                   cmsMeth   - CMS method key
c                                               = 0 - fixed interface method
c                                               = 1 - free interface method
c                                               = 3 - residual-flexible free interface method
c                                               = 4 - user defined method
c                                   cmsMixF   - flags mixed interface method derived from
c                                               = 1 - free interface method
c                                               = 3 - Residual-flexible free interface method
c                                               = 0 - default
c                                   hrmopt     - harmonic analysis type
c                                                = 0 Not a harmonic analysis
c                                                > 0 Harmonic analysis type
c                                   ActFlag    - active element logic flag for the two-step method (only assemble
c                                                - equivalent to actSymbFlag defined in soptcm.inc
c                                                - currently, it is used for AM and target element trimming
c                                                  = 2 - mark active nodes in casiInitialize before assembly due to
c                                                  = 1 - mark active nodes in casiInitialize before assembly due to
c                                                  = 0 - default
c                                   InActNKey  - sum of number of inactive nodes across all domain.
c                                                - it is only useful when ActFlag = 2
c                                                  = 0 - number of active nodes (nNodes) is equal to original ANSYS
c                                                      - there is no need to do AllToAll communication for the inac
c                                                  > 0 - number of active nodes (nNodes) is smaller than original A
c                                                      - if distributed full file is used, AllToAll communication a
c                                                      - back to ANSYS dof space
c                                   IANSndCnt  - total number of inactive nodes that need to be sent to other dom
c                                                - it is only usefull when InActNKey > 0 and distributed full file
c                                                  = 0 - no inactive nodes information in current domain that need
c                                                  > 0 - length of InActNodesSnd array
c                                   IANRcvCnt  - total number of inactive nodes that need to be received from oth
c                                                - it is only usefull when InActNKey > 0 and distributed full file
c                                                  = 0 - no inactive nodes information in current domain that need
c                                                  > 0 - length of InActNodesRcv array
c                                   Glblenbac  - Global number of nodes in ANSYS space
c                                   ptrActl,h  - pointer to act nodes to ANSYS nodes mapping array and other supp
c                                   ext_nNods  - nNodes + extra number of nodes needed by local equation based ma
c                                   ext_neqn   - neqn on file + extra number of equations needed by local equatio
c                                   ext_nmrow  - ext_neqn - nBC - number of BC in extra number of equations neede
c                                   EqnMtxFlag - flag indicate whether we are using equation based matrix in full
c                                                = 0 - element based full file (default)
c                                                = 1 - equation based full file
c                                   krysub     - key signaling that this .full file was created during the stage
c                                                the subspace for the KRYLOV method in a harmonic or transient an
c                                   ext_nce    - ncefull + extra number of CE/CP from extra number of equations n
c                                   ptrExtNl,h - pointer to extra dof mapping array
c                                   ExtSndCnt  - total number of extra equation that need to be sent to other dom
c                                   ExtRcvCnt  - total number of extra equation that need to be received from oth
c
c
c  ---      i       1      numdof   Degrees of freedom per node
c                                   DOF reference numbers are:
c         UX  = 1, UY  = 2, UZ  = 3, ROTX= 4, ROTY= 5, ROTZ= 6, AX  = 7, AY  = 8
c         AZ  = 9, VX  =10, VY  =11, VZ  =12, GFV1=13, GFV2=14, GFV3=15, WARP=16
c         CONC=17, HDSP=18, PRES=19, TEMP=20, VOLT=21, MAG =22, ENKE=23, ENDS=24
c         EMF =25, CURR=26, SP01=27, SP02=28, SP03=29, SP04=30, SP05=31, SP06=32

c  ---      i       1      lenbac   Nodal equivalence table. This table equates
c                                   the number used for storage to the actual
c                                   node number

c  ---      i       1   Glblenbac   Global nodal equivalence table.  This record
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                  EXISTS ONLY in the full file of the master
c                                  domain and ONLY for distributed .full files
c
c Stiffness Matrix. The next two records are repeated as a group neqn times.
c The pair of records will repeat GlbnVars times when model has across CEs.
c And row indices are global in the case.

c  STF      i       1      varies   Matrix row indices. The last item
c                                  corresponds to the diagonal. The
c                                  length of this record will vary (actual
c                                  length is returned from routine BINRD8)

c  ---   dp/cmp     1      varies   Matrix terms

c                                  If keyuns=0, this record will contain the
c                                  terms before the diagonal.

c                                  If keyuns=1, this record will contain the
c                                  entire row.
c
c Load Vector

c  RHS   dp/cmp     1        neqn   Load vector terms.

c Imaginary part of Load Vector

c  IRHS    dp       1        neqn   Imaginary load vector terms.
c                                  This record EXISTS ONLY if its pointer in the header
c                                  is not zero. Record length will be GlbnVars for model
c                                  with across CE.

c Stiffness matrix diagonal vector

c  DIAGK   dp/cmp   1        neqn   diagonal vector data for stiffness matrix. Its length
c                                  will be GlbnVars for model with across CE.

c Stiffness matrix diagonal scaling vector

c  SCLK    dp/cmp   1    ext_neqn   diagonal scaling vector for stiffness matrix. Record length
c                                  will be GlbnVars for across CE model.

c DOF marker array

c  MRK      i       1        neqn   marker array flagging various types of DOF
c                                  (1=U_EQN, 2=P_EQN, 3=E_EQN, 4=A_EQN).  Positive
c                                  values mean the DOF belongs to a user-defined node,
c                                  negative values mean the DOF belongs to an internal node.
c                                  NOTE: if this array does not exist then it is assumed that
c                                  all DOFs are U_EQNs for user-defined nodes.
c                                  Record length will be GlbnVars for models with across CE.

c PHYSICS marker array

c  PHYS     i       1        neqn   marker array flagging the various types of physics
c                                  (1=STRUCTURAL_EQN, 2=THERMAL_EQN, 3=ELECTRICAL_EQN,
c                                   4=MAGNETIC_EQN, 5=FLUID_EQN, 6=DIFFUSION_EQN)
c                                  NOTE: if this array does not exist then it is assumed that
c                                  all DOFs are STRUCTURAL_EQNs.
c                                  Record length will be GlbnVars for models with across CE.

c DOF information

c  DOF      i       1   ext_nNods   Nodal extent vector. Number of DOFs at each node

c  ---      i       1   GlbnNodes   Global nodal extent vector giving numbers of DOFs at each
c                                  global nodes. This record EXISTS ONLY for models using DMP
c                                  with across CE.

c  ---      i       1   GlbnNodes   A vector mapping global node number to local node number.
c                                  -1 in the vector means the node is not in this domain.
c                                  This record EXISTS ONLY for model with across CE.
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

```
c  ---      i       1    ext_neqn   DOF vector. If negative, this DOF is constrained.

c  ---      i       1    GlbnVars   A vector of global DOF reference numbers.
c                                   this record EXISTS ONLY for models using DMP with across CE.

c  ---      i       1   ext_nNods   A vector mapping local node number to global node number.
c                                   This record EXISTS ONLY for distributed .full files

c act nodes to ANSYS nodes mapping array

c  ---      i       1    ext_nNods  A vector mapping active nodes used in assembly to ANSYS nodes.

c  ---      i       1    GlbnNodes  A vector mapping global active nodes to global ANSYS nodes
c                                   This record EXISTS ONLY for models using DMP with across CE.

c  ---      i       1      numCPU   A vector (InActNumNodesSnd) giving the number of inactive nodes
c                                   that needs to be sent to each domain
c                                   This record EXISTS ONLY for distributed .full files and InActNKey > 0

c  ---      i       1      numCPU   A vector (InActNumNodesRcv) giving the number of inactive nodes
c                                   that needs to be received from each domain
c                                   This record EXISTS ONLY for distributed .full files and InActNKey > 0

c  ---      i       1   IANSndCnt   A vector (InActNodesSnd) giving local internal node number that needs to be
c                                   sent to other domains.
c                                   This record EXISTS ONLY when IANSndCnt > 0

c  ---      i       1   IANRcvCnt   A vector (InActNodesRcv) giving local internal node number that needs to be
c                                   received from other domains.
c                                   This record EXISTS ONLY when IANRcvdCnt > 0

c extra equations mapping array

c  ---      i       1      numCPU   A vector (extend_neqns_snd) giving the number of extra equations
c                                   that needs to be sent to each domain
c                                   This record EXISTS ONLY for distributed .full files and EqnMtxFlag == 1

c  ---      i       1      numCPU   A vector (extend_neqns_rcv) giving the number of extra equations
c                                   that needs to be received from each domain
c                                   This record EXISTS ONLY for distributed .full files and EqnMtxFlag == 1

c  ---      i       1   ExtSndCnt   A vector (extend_eqn_snd giving local equation/dof number that needs to be
c                                   sent to other domains.
c                                   This record EXISTS ONLY when ExtSndCnt > 0

c  ---      i       1   ExtRcvCnt   A vector (extend_eqn_rcv) giving local equation/dof number that needs to be
c                                   received from other domains.
c                                   This record EXISTS ONLY when ExtRcvCnt > 0

c BC information

c  BC       i       1        neqn   DOFs with imposed values

c  ---   dp/cmp     1      varies   Imposed values

c Mass Matrix.
c    if lumpm = 0:
c      The next two records are repeated as a group neqn times.
c      It will be in global form the same way as stiffness matrix if model has across CE.

c  MAS      i       1      varies   Matrix row indices. The last item
c                                   corresponds to the diagonal. The
c                                   length of this record will vary (actual
c                                   length is returned from routine BINRD8)

c  ---     dp       1      varies   Matrix terms

c    if lumpm = 1:
c  ---     dp       1        neqn   Matrix diagonals.
c                                   Record length will be GlbnVars for across CE model.
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

```
c Mass matrix diagonal vector

c  DIAGM   dp       1        neqn   diagonal vector data for mass matrix.
c                                   Record length will be GlbnVars for across CE model.

c Damping Matrix. The next two records are repeated as a group neqn times.
c For model with across CE, it will be in global form the same way as stiffness matrix.

c  DMP      i       1      varies   Matrix row indices. The last item
c                                   corresponds to the diagonal. The
c                                   length of this record will vary (actual
c                                   length is returned from routine BINRD8)

c  ---     dp       1      varies   Matrix terms

c Damping matrix diagonal vector

c  DIAGC   dp       1        neqn   diagonal vector data for damping matrix.
c                                   Record length will be GlbnVars for across CE model.

c K complex Matrix. The next two records are repeated as a group neqn times.
c For model with across CE, it will be in global form the same way as stiffness matrix.

c  KC      i       1       varies   Matrix row indices. The last item
c                                   corresponds to the diagonal. The
c                                   length of this record will vary (actual
c                                   length is returned from routine BINRD8)

c  ---     dp       1      varies   Matrix terms


c Nodal graph vectors

c  NGPH     i       1      nNodes   number of nonzeroes for each node.
c                                   Record length will be GlbnNodes for across CE model

c   Repeat for each node

c           i       1      varies   Index vector. Node number in the vector is global when
c                                   model has across CE

c G matrix if ncefull > 0.

c  CE       i       1     ncefull   List of dependent DOFs of local CEs.
c                                   It EXISTS ONLY if ncefull>0. The dependent DOF is local

c  ---      i       1      lcAcCE   List of dependent DOFs of local across CEs.
c                                   This record EXISTS ONLY if lcAcCE>0. The dependent DOF is
c                                   local

c  ---      i       1    GlbfAcCE   List of dependent DOFs of all across CEs.
c                                   This record EXISTS ONLY if GlbfAcCE>0. And it is ONLY
c                                   in the full file of master domain

c  ---      i       1      GlbfCE   List of dependent DOFs of all CEs. This record EXISTS ONLY
c                                   if GlbAcCE>0.  And it is ONLY in the full file of the
c                                   master domain

c  ---     dp       1     ncefull   g vector (constant terms) of local CEs. This record
c                                   EXISTS ONLY if ncefull>0

c  ---     dp       1     ncefull   imaginary g vector (constant terms) of local CEs. This
c                                   vector only exists for FULL harmonic analyses (kan=3).
c                                   This record EXISTS ONLY if ncefull>0

c  ---     dp       1     ncefull   g vector (constant terms) of local CEs for nonlinear
c                                   analysis predictor logic.
c                                   This record EXISTS ONLY if ncefull>0 & predKey=1.

c  ---     dp       1    GlbfAcCE   g vector (constant terms) of across CEs. This record
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                   EXISTS ONLY if GlbfAcCE>0 in the full file of the
c                                   master domain

c  ---     dp       1    GlbfAcCE   imaginary g vector (constant terms) of across CEs. This
c                                   vector only exists for FULL harmonic analyses (kan=3).
c                                   This record EXISTS ONLY if GlbfAcCE>0 in the full file
c                                   of the master domain

c  ---     dp       1    GlbfAcCE   g vector (constant terms) of across CEs for nonlinear
c                                   analysis predictor logic.  This record EXISTS ONLY
c                                   if GlbfAcCE>0 & predKey=1 in the master full file.

c  Following local CE data records EXIST ONLY in the full file of the domain with local CEs:
c  ---      i       1           4   Header for local CEs; 1=nRows, 2=nRows, 3=1, 4=0

c  ---      i       1       nRows   Vector of 1's

c  ---      i       1       nRows   Number of non-zero terms in each row for one local CE

c   Repeat for each row:

c  ---      i       1      varies   Column indices in local equation numbers

c  ---     dp       1      varies   Column values


c   Following across CE data records EXIST ONLY if GlbfAcCE>0 in the full file of master domain:
c  ---      i       1           4   Header for across CEs; 1=nRows, 2=nRows, 3=1, 4=0

c  ---      i       1       nRows   Vector of 1's

c  ---      i       1       nRows   Number of non-zero terms in each row for an across CE

c   Repeat for each row:

c  ---      i       1      varies   Column indices in global equation numbers

c  ---     dp       1      varies   Column values


c   Following CE data records EXIST ONLY if n>0 in the full file, where n == ncefull with
c   SMP and n == GlbfCE with DMP.  NOTE: for DMP these records only exist in the .full file
c   of master domain

c  ---      i       1           n   List of dependent DOFs of all CEs. This record EXISTS ONLY
c                                   if n>0.  And it is ONLY in the full file of the master
c                                   domain

c   NOTE: this matrix includes boundary d.o.f. which touch the CEs (for cnvfor.F)
c  ---      i       1           4   Header for across CEs; 1=nRows, 2=nRows, 3=1, 4=0

c  ---      i       1       nRows   Vector of 1's

c  ---      i       1       nRows   Number of non-zero terms in each row for an across CE

c   Repeat for each row:

c  ---      i       1      varies   Column indices in global equation numbers

c  ---     dp       1      varies   Column values




c Virtual constraint vector

c  VBC      i       1    nVirtBCs   marker array (bit 1 set -> constrained DOF for residual vector)
c                                                (bit 2 set -> constrained DOF for enforced motion)
c                                                (bit 3 set -> eliminated DOF for substructure master DOF)
c           i       1    nVirtBCs   virtual constraint DOFs
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

```
c  GVBC     i       1    GnVirtBCs  marker array (bit 1 set -> constrained DOF for residual vector)
c                                              (bit 2 set -> constrained DOF for enforced motion)
c                                              (bit 3 set -> eliminated DOF for substructure master DOF)
c                                  this record EXISTS ONLY for models using DMP with across CE

c           i       1    GnVirtBCs  virtual constraint DOFs
c                                  this record EXISTS ONLY for models using DMP with across CE

c Substructuring master DOF vector

c  MDF      i       1    nMastDOF   list of master DOFs for substructuring

c  GMDF     i       1    GlbnMast   list of master DOFs for substructuring
c                                  this record EXISTS ONLY for models using DMP with across CE


c  Meaning of K11, K12, and G matrices:
c     Given
c         [K]{x} = {F}
c     subject to the constraints
c           {x1} = [G]{x2} + {g}
c     where {x1} are the dependent DOFs, {x2} the independent DOFs

c     This results in
c         [K*]{x2} = {F*}
c     where
c         [K*] = [G]'[K11][G] + [G]'[K12] + [K21][G] + [K22]
c         {F*} = [G]'{f1} + {f2} - [G]'[K11]{g} - [K21]{g}

c complex version of {F*} decomposed into, we assume G' is always real
c and g could be complex denoted as g' == (g,gx) :
c         G' K11' g' = G' (K11,M11)*(g,gx)
c                    = G' [K11*g - M11*gx, M11*g + K11*gx]

c         K21' *g'   = (K21,M21)*(g,gx)
c                    = (K21*g- M21*gx, K21*gx + M21*g)
```

1.10. Description of the Substructure Displacement File

This section explains the contents of the substructure displacement file (`jobname.dsub`).

1.10.1. Standard Mechanical APDL File Header

See The Standard Header for Mechanical APDL Binary Files (p. 4) for a description of this set. File number (Item 1) is 13.

1.10.2. DSUB File Format

```
*comdeck,fddsub

c *** Copyright ANSYS.  All Rights Reserved.
c *** ansys, inc.

c     **********  description of substructure displacement file  ****

      character*8  DSUBNM
      parameter   (DSUBNM='dsub    ')

      LONGINT         dsubfpL, dsubfp
      integer         dsubbk, dsubut
      common /fddsub/ dsubfpL, dsubbk, dsubut
```

```
      equivalence  (dsubfp,dsubfpL)

c     ***********************************************************
c     *                        CAUTION                          *
c     *   Please update  proc getDSUBInfo in SEManagement.eui   *
c     *   if the file format changes or GUI read of the DSUB    *
c     *   file will fail.                                       *
c     ***********************************************************

c   ********** common variable descriptions ***********
co  dsubfpL      file position on file dsub
co  dsubbk       block number for file dsub
co  dsubut       file unit for file dsub

c  open:   slvstr
c  write:  setdis,eostrt,eofini,ranbasPCG,ranbwvPCG,slvstr,slvend,supsr1,supidx,supscl
c  close:  slvend
c  read:   matstr as .usub


c   See fddesc for documentation of how binary files are stored.
c
c     **********  file format  **********

c        recid tells the identifier for this record.  Not all records will have
c             identifiers -- they are only indicated for those records whose
c             record pointers are stored in the second file header.

c        type tells what kind of information is stored in this record:
c             i - integer
c             dp - double precision
c             cmp - complex

c        nrec tells how many records of this description are found here

c        lrec tells how long the records are (how many items are stored)

c recid    type    nrec    lrec     contents

c  ---      i       1      100      standard ANSYS file header (see binhed8 for
c                                   details of header contents)

c   1       i       1       20      .DSUB FILE HEADER

c                                  fun13, fpeofS, fpeofL,    kcxp,  nmode,
c                                   knum,  kCXFM, senres,  cpxeng,      0,
c                                      0,      0,      0,       0,      0,
c                                      0,      0,      0,       0,      0,
c
c                           fun13   - file unit number
c                    fpeofS, fpeofL  - pointer to the eof
c                           kcxp    - = 1 if complex results
c                           nmode   - number of extracted modes if use pass modal
c                           knum    - number of expanded modes if use pass modal
c                           senres  - = 1: expand only displacements
c                       (all type of analysis)
c                                     = 2: expand real and then imaginary part
c                                          of displacements (harmonic analysis)
c                                     = 3: expand displacements, velocities,
c                                          accelerations (transient analysis)
c                              cpxeng  - = 1: after use pass = harmonic or modal damp
c                                          compute average, amplitude, peak energies

c      *** these records are repeated each iteration ***
c   2       i       1       50     fun13,    kan, lenbac,  numdof,   kcmplx,
c                                  itime,  itter, ncumit,  nitter,curdof(i),  (10)
c                                (curdof(i),i=1,numdof)
c                                (curdof(i),i=1,numdof)                     (20)
c                                (curdof(i),i=1,numdof)
c                                (curdof(i),i=1,numdof)                     (30)
c                                (curdof(i),i=1,numdof)
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                (curdof(i),i=1,numdof)                      (40)
c                              curdof(i),      0,      0,       0,        0,
c                                       0, extopt,qrDmpKy,Glblenbac,  timint,  (50)
c
c                             extopt  - mode extraction option
c                             qrDmpKy - QR damped calculations key

c   3      dp       1       20    time/freq,acel(1),acel(2),acel(3), frqenr,
c                                   frqud,      0,      0,      0,      0,
c                                       0,      0,      0,      0,      0,
c                                       0,      0,      0,      0,      0,

c      *** the following records are repeated for each superelement ***
c   4       i       1       20    iel,    nrow,   nvect,  ntrans, name1,
c                                 name2,  trok,   lrok,   name3,  name4,
c                                 name5,  name6,  name7,  name8,  kCXFM,
c                                 kdamp,      0,      0,      0,      0,
c
c                             iel    -   (iel=0 signals end of superelements
c                                       for this iteration)
c                             nrow   - number of dofs
c                             nvect  - number of load vectors
c                             ntrans - number of transformations
c                             name*  - name1/2/3/4/5/6/7/8 are the
c                                       inexc4 representation of the
c                                       32 character filename.
c                                       name3/4/5/6/7/8 will be "0"
c                                       for pre rev 5.2 files - cwa
c                             trok   - flag if transformations can be applied
c                             lrok   - flag if large deformation transformation
c                                       can be applied
c                             kCXFM  - key if CXFM transformations are present on file
c                             kdamp  - key if ratios:
c                                       rdamp(1),rdamp(2),dmprat,dmpst,spin,mscalm
c                                       are present on file
c
c
c   5      dp       1       125   ntrans sets of transformations 15x5
c   6      dp       1       250   ntrans sets of transformations (CS) 50x5 - present if kCXFM = 1
c   7     LONG      1      nrow   (lL(i),i=1,nrow) - global dofs
c                                  (made LONGINT in version 14.0)
c   8      dp       1     nvect   (scalf(i),i=1,nvect)
c   9      dp       1        10    rdamp(1),rdamp(2),dmprat,dmpstr,spin,   - present if kdamp = 1
c                                  mscalm  ,       0,     0,     0,   0,
c  10      dp       1      nrow   (disp(i),i=1,nrow)
c  11      dp       1      nrow   (vel(i),i=1,nrow)                        - present if senres >= 2
c  12      dp       1      nrow   (acel(i),i=1,nrow)                       - present if senres = 3
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 2: Accessing Binary Data Files

This chapter explains the routines you need to read, write, or modify a Mechanical APDL binary file. This collection of routines (called BINLIB) resides on your product-distribution media.

The following topics are discussed in this chapter:

2.1. Accessing Mechanical APDL Binary Files

The BINLIB library is in the dynamic link library `\Program Files\ANSYS Inc\V212\ANSYS\custom\misc\<platform>\binlib.dll` on Windows systems (where *<platform>* is a directory that uniquely identifies the hardware platform version) or the shared library `/ansys_inc/v212/ansys/customize/misc/<platform>/libbin.so` on Linux systems.

2.1.1. Access Routines for Results Files

Demonstration programs that use the BINLIB library for reading and writing Mechanical APDL results files are included on the installation media:

- `ResRdDemo`
- `ResWrDemo`
- `bintst`

On Windows Systems:

The FORTRAN source for these programs is located in `\Program Files\ANSYS Inc\V212\ANSYS\customize\user`. The files are named `ResRdDemo.F` and `ResWrDemo.F`.

To link these demonstration programs, use the `\Program Files\ANSYS Inc\V212\ANSYS\custom\misc\<platform>\rdrwrt.bat` procedure file and specify the program that you want to build on the command line. Valid command line options are `ResRdDemo`, `ResWrDemo`, and `userprog`. For example, to build the program to read a results file, type:

```
\Program Files\ANSYS Inc\V212\ANSYS\custom\misc\<platform>\rdrwrt  ResRdDemo
```

Appropriate files are then copied from `\Program Files\ANSYS Inc\V212\ANSYS\customize\user` to your working directory, compiled, and linked. The resulting executable is also placed in your current working directory.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Use the userprog command line option when writing your own customized program, naming the routine userprog.F. The resulting executable is named userprog.exe. When userprog is used, no files are copied from \Program Files\ANSYS Inc\V212\ANSYS\customize\user to your working directory.

These files are loaded onto your system only if you performed a custom installation and chose to install the customization tools.

On Linux systems:

The FORTRAN source for these programs is located in /ansys_inc/v212/ansys/customize/misc. The files are named ResRdDemo.F and ResWrDemo.F.

To link these demonstration programs, use the /ansys_inc/v212/ansys/customize/misc/rdrwrt.link procedure file and specify the program that you want to build on the command line. Valid command line options are ResRdDemo, ResWrDemo, and userprog. For example, to build the program to read a results file, type:

```
/ansys_inc/v212/ansys/customize/misc/rdrwrt.link  ResRdDemo
```

Appropriate files are then copied from /ansys_inc/v212/ansys/customize/misc to your working directory, compiled, and linked. The resulting executable is also placed in your current working directory. Procedure files are available in the /ansys_inc/v212/ansys/bin directory to run these programs, once linked. The procedure files are named ResRdDemo212 and ResWrDemo212.

Use the userprog command line option when writing your own customized program, naming the routine userprog.F. The resulting executable is named userprog.e212. When userprog is used, no files are copied from /ansys_inc/v212/ansys/customize/misc to your working directory. The procedure file is named userprog212.

These files are loaded onto your system only if you performed a custom installation and chose to install the customization tools.

2.1.2. Characteristics of Mechanical APDL Binary Files

Before accessing Mechanical APDL binary files, you need to know certain file characteristics:

1. A Mechanical APDL binary file is a direct-access, unformatted file. You read or write a record by specifying (as a number) what location to read or write.

2. Before Mechanical APDL actually writes data to a file on a disk, it uses buffers to store data in memory until those buffers become full. A block number designates these buffers. Most access routines use this block number.

3. By default, Mechanical APDL files are external files. The standardized "external" format the files use enables you to transport them across different computer systems.

4. In addition to file names, Mechanical APDL uses file numbers to identify the files. File handles and other information are associated with the file numbers.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Some binary files contain data values that point to the start of certain data (for example, the start of the data steps index table record). Both Mechanical APDL and external binary files access routines use these pointers to locate data on the various binary files.

6. All data is written out as 32-bit integers. Double-precision data and pointers, therefore, take up two integer words. To create a 64-bit pointer from the two 32-bit integers, use the function largeIntGet.

2.1.3. Viewing Binary File Contents

To view the contents of certain Mechanical APDL binary files, you issue the command **/AUX2** or choose menu path **Utility Menu>File>List>Binary Files** or **Utility Menu>List>File>Binary Files**. (You can do so only at the Begin level.) Mechanical APDL then enters its binary file dumping processor, AUX2, and dumps the binary file record by record.

In AUX2, you can use either the record number (**DUMP** command) or the record pointer (**PTR** command). If the file was written in parallel (-NP>1 on the command line), the **DUMP** command may not work as expected. In that case, only the **PTR** command may be used.

2.1.4. Abbreviations

The input and output for the routines discussed in this chapter are described with the following abbreviations:

- *Type* of variable is one of the following:

  int - integer (4-byte)
  long - integer (8-byte)
  dp - double-precision
  log - logical (true or false)
  char - character
  comp - double precision complex

- *Size* of variable is one of the following:

  sc - scalar variable
  ar(*n*) - array size *n*
  func - functional return value

- *Intent* of variable is one of the following:

  in - input only
  out - output only
  inout - both an input and an output variable

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.1.5. binini (Initializing Buffered Binary I/O Systems)

```
*deck,binini
      subroutine binini (iott)
c *** primary function: initialize buffered binary i/o system
c --- This routine is intended to be used in standalone programs.
c --- This routine should not be linked into the ANSYS program.

c *** Notice - This file contains ANSYS Confidential information ***

c  input arguments:
c     iott      (int,sc,in)       - output unit number for error output

c  output arguments:   none
```

2.1.6. Function sysiqr (Retrieving the Status of a File)

```
*deck,sysiqr
      function sysiqr (nunit,fname,lname_in,inqr_in)

c *** primary function: do a file system inquire (system dependent)

c *** Notice - This file contains ANSYS Confidential information ***

c  input arguments:
c    variable (typ,siz,intent)    description
c    nunit    (int,sc,in)       - fortran unit number (used only for inqr='O')
c    fname    (chr,sc,in)       - name of file
c    lname_in (int,sc,in)       - length of file name (characters, max=50)
c    inqr_in  (chr,sc,in)       - character key for information requested
c                                  = 'E' - return whether file exists
c                                      sysiqr = 1 - file exists
c                                             = 0 - file does not exist
c                                             < 0 - error occured
c                                  = 'O' - return whether file is open
c                                      sysiqr = 1 - file is open
c                                             = 0 - file is closed
c                                             < 0 - error occured
c                                  = 'N' - return unit number of file
c                                      sysiqr > 0 - unit number for file
c                                             = 0 - file not assigned to a unit
c                                             < 0 - error occured

c  output arguments:
c    sysiqr   (int,func,out)    - the returned value of sysiqr is based on
c                                   setting of inqr
```

2.1.7. Function biniqr8 (Retrieving System-Dependent Parameters)

```
*deck,biniqr8
      function biniqr8 (nblk,key)
c *** primary function: get data about a block i/o buffer
c --- This routine is intended to be used in standalone programs.
c --- This routine should not be linked into the ANSYS program.

c *** Notice - This file contains ANSYS Confidential information ***

c  input arguments:
c     nblk      (int,sc,in)       - the block number for the inquiry
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                    or zero (see below)
c     key       (int,sc,in)       - key for information requested
c                        nblk = 0 - return information about system/file
c                                 key = 1 - return system block size
c                                     = 2 - return number of integers per dp
c                                     = 3 - return filename length
c                                       5 = return integers per LONG
c                        nblk > 0 - return information about this block
c                                 key = 1 - return fortran unit number
c                                     = 2 - return number of pages in file
c                                     = 3 - return length of page (32 bit words)
c                                     = 4 - return open status
c                                           0 - file close
c                                           1 - file open
c                                     = 5 - return file format
c                                           0 - internal format
c                                           1 - external format
c                                     = 6 - return read/write status
c                                           0 - both read & write
c                                           1 - read
c                                           2 - write
c                                     = 7 - return current position on file
c                                     = 8 - return maximum length of file
c                                             (in words)
c                                     = 9 - return starting word for this page
c                                             in buffer
c                                     =10 - return base location
c                                     =11 - return debug key
c                                     =12 - return absolute (non-base) key
c                                     =15 - return max record written
c                                     =16 - return swap and record header key
c                                     =17 - return precision key

c  output arguments:
c     biniqr8   (LONG,func,out)   - the returned value of biniqr is based on
c                                       setting of nblk and key
```

2.1.8. Function binset (Opening a Blocked Binary File or Initializing Paging Space)

```
*deck,binset
      function binset (nblk,nunit,ikeyrw,istart,paglen,npage,pname,
     x                 nchar,kext,Buffer4)

c *** primary function: initialize paging space for a blocked binary file.
c                       binset should be used to open a blocked file
c                       before binrd8 or binwrt8 are used.  binclo should
c                       be used to close the file.
c --- This routine is intended to be used in standalone programs.
c --- This routine should not be linked into the ANSYS program.

c *** Notice - This file contains ANSYS Confidential information ***

c  input arguments:
c     nblk      (int,sc,in)       - block number (1 to BIO_MAXFILES max)
c     nunit      (int,sc,in)      - fortran unit number for the file
c                                   (if 0, bit bucket)
c     ikeyrw    (int,sc,in)       - read/write flag
c                                   =  0 - both read & write
c                                   =  1 - read
c                                   =  2 - write
c                                   =  9 - read only
c       NOTE: 0 may write, but the file length may not be extended and
c               the file may or may not exist
c             1 reads only, but the file protection must set set to "rw"
```

```
c             2 may extend the file length and the file is a new file
c             9 reads only, but the file protection may be "r" only
c     istart    (int,sc,in)       - starting location in buffer array
c                                   usually 1 for nblk=1, paglen*npage+1
c                                   for nblk=2,etc.
c     paglen    (int,sc,in)       - page length in integer*4 words for external
c                                   files
c                                   paglen should always be a multiple of
c                                   512 words for efficiency
c     npage     (int,sc,in)       - number of pages (1 to BIO_MAXBLOCKS max)
c     pname     (chr,ar(*),in)    - name of the file
c     nchar     (int,sc,in)       - number of characters in the file name (not
c                                   used)
c     kext      (int,sc,in)       - no longer used, always external format
c     Buffer4   (i4, ar(*),inout) - work array for paging, should be
c                                   dimensioned to paglen*npage*nblk (max)

c  output arguments:
c     binset    (int,func,out)    - error status
c                                   = 0 - no error
c                                   <>0 - error occurred
c     Buffer4   (i4, ar(*),inout) - work array for paging
```

2.1.9. Subroutine bintfo (Defining Data for a Standard Mechanical APDL File Header)

```
*deck,bintfo
      subroutine bintfo (title,jobnam,units,code)
c *** primary function:    set information necessary for binhed8
c --- This routine is intended to be used in standalone programs.
c --- This routine should not be linked into the ANSYS program.

c
c *** Notice - This file contains ANSYS Confidential information ***
c
c     typ=int,dp,log,chr,dcp   siz=sc,ar(n)   intent=in,out,inout
c
c  input arguments:
c     variable (typ,siz,intent)    description
c     title    (chr*80,ar(2),in) - main title and 1st subtitle
c     jobnam   (chr*8,sc,in)     - jobname
c     units    (int,sc,in)       - units
c                                    = 0 - user defined units
c                                    = 1 - SI
c                                    = 2 - CSG
c                                    = 3 - U.S. Customary, using feet
c                                    = 4 - U.S. Customary, using inches
c                                    = 5 - MKS
c                                    = 6 - MPA
c                                    = 7 - uMKS
c     code     (int,sc,in)       - code defining 3rd party vendor
c                                   (contact ANSYS, Inc. for code assignment)
c
c  output arguments:
c     none
c
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

2.1.10. Subroutine binhed8 (Writing the Standard Mechanical APDL File Header)

```
*deck,binhed8
      subroutine binhed8 (nblk,nunit,filposL,Buffer4)
c *** primary function:    put standard header on a binary file, all
c                          permanent binary files should have this header
c *** secondary functions: return the first data position

c --- This routine is intended to be used in standalone programs.
c --- This routine should not be linked into the ANSYS program.

c *** Notice - This file contains ANSYS Confidential information ***

c  input arguments:
c     nblk     (int,sc,in)        - block number of open binary file
c                                   (as defined with subroutine binset)
c     nunit    (int,sc,in)        - the unit number for this file
c     Buffer4  (int,ar(*),inout)  - work array for paging, should be the
c                                   same array as used in binset

c  output arguments:
c     filposL  (LONG,sc,out)      - the position after the header
c     Buffer4  (int,ar(*),inout)  - work array for paging

c   **********  ANSYS standard header data description (100 words)  **********
c  loc   no. words   contents
c   1       1        fortran unit number
c   2       2        file format
c                    = 0 - old internal format
c                    = -1 - external IEEE format
c                    = 1  old external format
c   3       1        time in compact form (ie 130619 is 13:06:19)
c   4       1        date in compact form (ie 20051023 is 10/23/2005)
c   5       1        units
c                    = 0 - user defined units
c                    = 1 - SI
c                    = 2 - CSG
c                    = 3 - U.S. Customary, using feet
c                    = 4 - U.S. Customary, using inches
c                    = 5 - MKS
c                    = 6 - MPA
c                    = 7 - uMKS
c   6       1        User_Linked
c  10       1        revision in text format '10.0' (inexc4)
c  11       1        date of revision release for this version
c  12       3        machine identifier - 3 4-character strings
c  15       2        (obsolete - see below) jobname - 2 4-character strings
c  17       2        product name - 2 4-character strings
c  19       1        special version label - 1 4-character string
c  20       3        user name - 3 4-character strings
c  23       3        machine identifier - 3 4-character strings
c  26       1        system record size at file write
c  27       1        maximum file length
c  28       1        maximum record number
c  31       8        jobname - 8 4-character strings
c  41      20        main title - 20 4-character strings
c  61      20        first subtitle - 20 4-character strings
c  81       1        file compression level
c  82       1        file sparsification key
c  95       1        split point of file
c                     NOTE: Split files are not support by binlib!
c  96       1        reserved for future use
c 97-98     2        LONGINT of file size at write
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A version of binhed8 exists without the "8" suffix (binhed.F) that takes a regular integer for the third argument. This subroutine, therefore, cannot address large files where the file position exceeds 2**31. Use the binhed8.F version for any new program.

2.1.11. Subroutine binrd8 (Reading Data from a Buffered File)

```
*deck,binrd8
      subroutine binrd8 (nblk,LongLocL,leng,ivect,kbfint,Buffer4)

c **********  buffer read routine  **********

c --- This routine is intended to be used in standalone programs.
c --- This routine should not be linked into the ANSYS program.

c *** Notice - This file contains ANSYS Confidential information ***

c  input arguments:
c   nblk    (int,sc,in)      - block number.  see fd___(i.e. fdresu for results
c                              file, etc.) include decks for more information.
c   LongLocL(LONG,sc,inout)  - location in integer*4 words of the startin
c                              position on the file.
c   leng    (int,sc,inout)   - number of words to read into ivect. (must be
c                              less or equal to dimension given to ivect in
c                              the calling routine).  if ivect is to be used
c                              as integers, use as is.  if ivect is to be
c                              used for double precision numbers, it must be
c                              increased by multiplying it by INTPDP.
c                              if negative, skip record and do not return
c                              data(results).
c   Buffer4 (i4,ar(*),inout) - work array for paging, should be the
c                              same array as used in binset

c  output arguments:
c   LongLocL(LONG,sc,inout)  - location in integer*4 words of the current
c                              position on the file.  It is updated after
c                              each read operation
c   leng    (int,sc,inout)   - tells you how many items it actually read(in
c                              integer words).
c                              if zero, end of file(error case)
c   ivect   (int,ar(*),out)  - results (can be either integer or double
c                              precision in the calling routine)
c   kbfint  (int,sc,out)     - key for type(used only for AUX2 dump)
c                              = 0  double precision data
c                              > 0  integer data(usually the same as leng)
c   Buffer4 (i4,ar(*),inout) - work array for paging
```

Versions of binrd8/binwrt8 exist without the "8" suffix (binrd/binwrt) that take a regular integer for the second argument. These subroutines, therefore, cannot address large files where the file position exceeds 2**31. Use the binrd8/binwrt8 versions for any new programs.

2.1.12. Subroutine binwrt8 (Writing Data to a Buffered File)

```
*deck,binwrt8
      subroutine binwrt8 (nblk,LongLocL,leng,ivect,kbfint,Buffer4)

c *** primary function: buffer write routine

c --- This routine is intended to be used in standalone programs.
c --- This routine should not be linked into the ANSYS program.

c *** Notice - This file contains ANSYS Confidential information ***
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c  input arguments:
c   nblk    (int,sc,in)       - block number.  see fd___(i.e. fdresu for results
c                               file, etc.) include decks for more information.
c   LongLocL(LONG,sc,inout)   - location in integer words of the starting
c                               position on the file.
c   leng    (int,sc,in)       - number of words to read from ivect. (must be
c                               less or equal to dimension given to ivect in
c                               the calling routine).  if ivect is to be used
c                               as integers, use as is.  if ivect is to be
c                               used for double precision numbers, it must be
c                               increased by multiplying it by INTPDP.
c   ivect   (int,ar(*),in)    - data to be written onto the file(can be either
c                               integer or double precision in the calling
c                               routine)
c   kbfint  (int,sc,in)       - key for type(used only for AUX2 dump)
c                               = 0  double precision data
c                               > 0  integer data(usually the same as leng)
c   Buffer4 (int,ar(*),inout) - work array for paging, should be the
c                               same array as used in binset on this block

c  output arguments:
c   LongLocL(LONG,sc,inout)   - location in integer words of the current
c                               position on the file.  It is updated after
c                               each write operation
c   ivect   (int,ar(*),out)   - vector containing record to be written
c   Buffer4 (int,ar(*),inout) - work array for paging
```

Versions of `binrd8`/`binwrt8` exist without the "8" suffix (`binrd`/`binwrt`) that take a regular integer for the second argument. These subroutines, therefore, cannot address large files where the file position exceeds 2**31. Use the `binrd8`/`binwrt8` versions for any new programs.

2.1.13. Subroutine exinc4 (Decoding an Integer String into a Character String)

```
*deck,exinc4
      subroutine exinc4 (ichext,chin,n)
c primary function: decode externally formatted integer versions of 4-character
c                   strings to plain 4-character strings (used to convert data
c                   from externally formatted files to data for interally
c                   formatted files)
c
c *** Notice - This file contains ANSYS Confidential information ***
c
c  input arguments:
c     ichext   (int,ar(n),in)    - externally formatted integer form of
c                                  4-character strings
c     n        (int,sc,in)       - number of strings to convert
c
c  output arguments:
c     chin     (char,ar(n),out)  - strings in character form
```

2.1.14. Subroutine inexc4 (Coding a Character String into an Integer String)

```
*deck,inexc4
      subroutine inexc4 (chin,ichext,n)
c primary function: encode plain 4-character strings into externally formatted
c                   integer versions of 4-character strings (used to convert
c                   data from internally formatted files to data for
c                   externally formatted files)
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c
c *** Notice - This file contains ANSYS Confidential information ***
c
c  input arguments:
c     chin      (char,ar(n),in)   - strings in character form
c     n         (int,sc,in)       - number of strings to convert
c
c  output arguments:
c     ichext    (int,ar(n),out)   - externally formatted integer form of
c                                   4-character strings
```

2.1.15. Subroutine binclo (Closing or Deleting a Blocked Binary File)

```
*deck,binclo
      subroutine binclo (nblk,pstat,Buffer4)
c *** primary function: close a blocked file, every block/file opened with
c                       binset should be closed with binclo
c *** secondary function: the file can be deleted by specifying 'D' in pstat
c --- This routine is intended to be used in standalone programs.
c --- This routine should not be linked into the ANSYS program.

c *** Notice - This file contains ANSYS Confidential information ***

c  input arguments:
c     nblk      (int,sc,in)       - the block number to close
c                                   (as defined with subroutine binset)
c     pstat     (chr,sc,in)       - keep or delete flag
c                                   = 'K' - keep file
c                                   = 'D' - delete file
c     Buffer4   (int,ar(*),inout) - work array for paging, should be the
c                                   same array as used in binset

c  output arguments:
c     Buffer4   (int,ar(*),inout) - work array for paging
```

2.1.16. Subroutine largeIntGet (Converting Two Integers into a Pointer)

```
*deck,largeIntGet
      function largeIntGet (small,large)

c primary function:    Convert two short integers into a long integer

c object/library:  res

c *** Notice - This file contains ANSYS Confidential information ***

c  input arguments:
c     small        (int,sc,in)       - least significant part
c     large        (int,sc,in)       - most significant part

c  output arguments:
c     largeIntGet  (LONGINT,sc,out)  - 64 bit integer
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

2.2. Demonstration Routines

The demonstration routines demonstrate several ways to use the binary file access routines provided with Mechanical APDL. The programs described below (all available on your distribution media; see Accessing Mechanical APDL Binary Files (p. 69) for their location) demonstrate other tasks that the binary access routines can do.

2.2.1. Program bintst (Demonstrates Dumping a Binary File and Copying It for Comparison Purposes)

The `bintst` program dumps a binary file with the name `file.rst` to the screen. It then takes that file, copies it to a new file, `file2.rst`, and dumps the new file to the screen for comparison purposes.

2.2.1.1. Common Variables:

Variable	Type, Size, Intent	Description
iout	int, sc, comm	The output unit number
intpdp	int, sc, comm	The number of integers per double precision word
lenfnm	int, sc, comm	The number of characters in the filename
reclng	int, sc, comm	The system record length

Note:

The bintst program is not part of the `binlib.a` library. It is included here only to aid you.

2.2.2. Subroutine bintrd (Demonstrates Printing a Dump of File Contents)

```
*deck,bintrd
      subroutine bintrd (pname)
c *** primary function: bin file dump utility
c
c *** Notice - This file contains ANSYS Confidential information ***
c
c Copyright ANSYS.  All Rights Reserved.
c *** ansys, inc.
c
c     typ=int,dp,log,chr,dcp   siz=sc,ar(n)   intent=in,out,inout
c
c  input arguments:
c     variable  (typ,siz,intent)    description
c     pname     (chr,sc,in)       - name of binary file which is to
c                                   be dumped to the screen
c
c  output arguments:
c     none.
c
c  common variables:
c     iout      (int,sc,comm)     - output unit number
c     intpdp    (int,sc,comm)     - number of integers per double precision word
c     lenfnm    (int,sc,comm)     - number of characters in the filename
c     reclng    (int,sc,comm)     - system record length
c
c                           NOTE: bintrd is not part of binlib.a.  it is
```

```
c                                 included only as an aid to users.
c
```

Note:

The `bintrd` routine and the `bintwr` routine described below are not part of `binlib.a`. This chapter includes it only to aid you. You can find the source for this routine on the product-distribution media.

Both subroutines require the following common:

```
COMMON/BINTCM/ IOUT,INTPDP,LENFNM,RECLNG
```

- *`Iout`* is the output unit number.
- *`Intpdp`* is the number of integers per double precision word.
- *`Lenfnm`* is the number of characters in the filename.
- *`Reclng`* is the system record length.

2.2.3. Subroutine bintwr (Demonstrates Copying Binary File Contents)

```
*deck,bintwr
      subroutine bintwr (pname,nname)
c *** primary function: bin file copy utility
c
c *** Notice - This file contains ANSYS Confidential information ***
c
c Copyright ANSYS.  All Rights Reserved.
c *** ansys, inc.
c
c     typ=int,dp,log,chr,dcp   siz=sc,ar(n)   intent=in,out,inout
c
c  input arguments:
c     variable  (typ,siz,intent)    description
c     pname     (chr,sc,in)       - name of binary file which is to be copied
c
c  output arguments:
c     variable  (typ,siz,intent)    description
c     nname     (chr,sc,out)      - name of new binary file which is a copy
c                                   of pname
c  common variables:
c     iout      (int,sc,comm)     - output unit number
c     intpdp    (int,sc,comm)     - number of integers per double precision word
c     lenfnm    (int,sc,comm)     - number of characters in the filename
c     reclng    (int,sc,comm)     - system record length
c
c                           NOTE: bintwr is not part of binlib.a.  it is
c                                 included only as an aid to users.
c
```

2.2.4. Program ResRdDemo (Demonstrates Reading a Results File)

Program `ResRdDemo` demonstrates how to read a results file using the results file access routines (p. 90). The file must be named `test.rst` and the file contents are written to the screen.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This file resides in the subdirectory `\Program Files\ANSYS Inc\V212\ANSYS\customize\user` (on Windows systems) or `/ansys_inc/v212/ansys/customize/misc` (on Linux systems).

2.2.5. Program ResWrDemo (Demonstrates Writing a Results File)

Program `ResWrDemo` demonstrates how to write a Mechanical APDL-readable results file. This file resides in the subdirectory `\Program Files\ANSYS Inc\V212\ANSYS\customize\user` (on Windows systems) or `/ansys_inc/v212/ansys/customize/misc` (on Linux systems).

2.3. Retrieving Data from the Results File

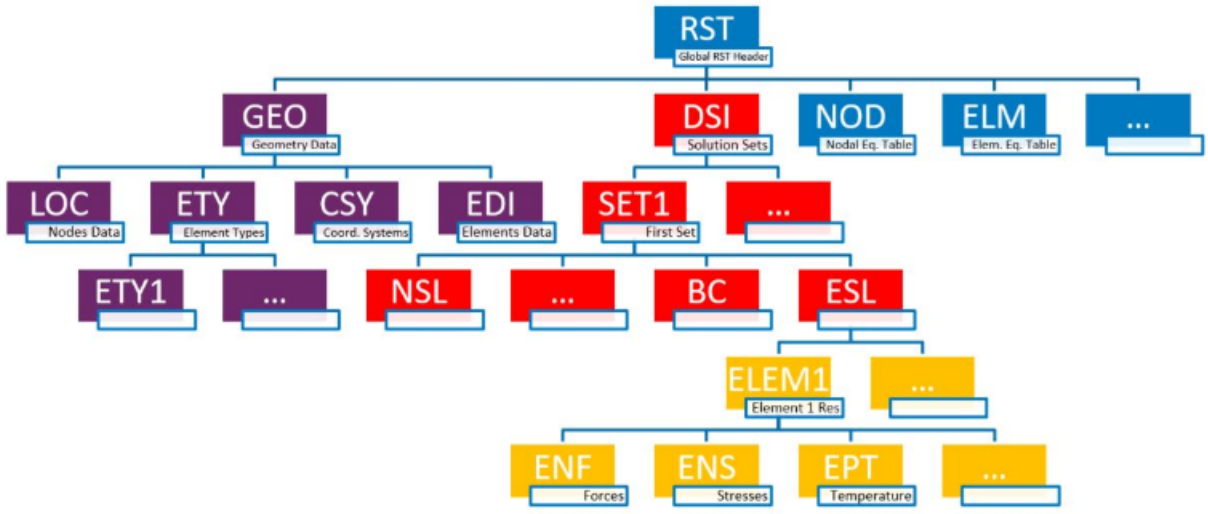
There are two methods for retrieving data from the results file:

2.3.1. Results File Reader

The Mechanical APDL solver results are typically stored in a binary file using the format described in Description of the Results File (p. 5). This file can have an `.rst`, `.rth`, or `.rmg` extension, depending on the analysis type.

The results file format can be visualized as a tree of records.

Figure 2.1: Results File Structure



RST File Structure

Color code: blue - general records; purple - geometry; red - results sets; yellow - element results

The Results File Reader is a C++ object API that facilitates access to these records.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can relate the above tree structure to the records described in Results File Format (p. 6). The below table shows typical examples.

Table 2.1: Finding Records in the Results File Description

Record Label	Meaning	Sample Content from `.rst` File

GEO	Geometry data	`c GEO i 1 80 Geometry data header(was 20 in 32 bit v`
		`c 0, maxety, maxrl, nnod, r`
		`c maxcsy, ptrETY, ptrREL, ptrLOC, ptr`
		`...`

DSI	Data set index for solution sets	`c DSI i 1 2*resmax Data sets index table. This record cont`
		`c the record pointers for the beginning o`
		`c each data set. The first resmax records`
		`c the first 32 bits of the index, the sec`
		`c resmax records are the second 32 bits.`
		`...`

NOD	Nodal equivalence table	`c NOD i 1 nnod Nodal equivalence table. This table equ`
		`c the number used for storage to the actu`
		`c node number`
		`c (Back(i),i=1,nnod)`

ELM	Element equivalence table	`c ELM i 1 nelm Element equivalence table. The program`
		`c stores all element data in the numerica`
		`c order that the SOLUTION processor solve`
		`c elements. This table equates the order`
		`c number used to the actual element numbe`

The following sections describe how to use the Results File Reader:

2.3.1.1. Compile and Link with the Result File Reader Code

To compile your code based on the Results File Reader, you must add the path that contains the main `C_RstFile.h` include file and its dependencies, as shown below.

Windows:

```
C:\Program Files\ANSYS Inc\v212\ansys\customize\include
```

Linux:

```
/ansys_inc/v212/ansys/customize/include/
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

On Windows, you must add the -DWINNT preprocessor definition at compile time. On Linux, you must add -DLINUX.

To link your application, add the following libraries to your link process.

Windows:

```
C:\Program Files\ANSYS Inc\v212\ansys\custom\lib\winx64\libCadoeKernel.lib

C:\Program Files\ANSYS Inc\v212\ansys\custom\lib\winx64\libCadoeReaders.lib
```

Linux:

```
/ansys_inc/v212/ansys/lib/linx64/libCadoeKernel.so

/ansys_inc/v212/ansys/lib/linx64/libCadoeReaders.so
```

Note that these libraries have dependencies to the Intel MKL library.

2.3.1.2. Open an Existing Results File

The first step is to add this line at the beginning of your C++ code:

```
#include "C_RstFile.h"
```

Suppose that you have a results file named `file.rst` in your current directory. Use this line to open an existing file:

```
C_RstFile     RstFile( "file.rst");   // File is opened. Ready to be parsed.
```

The file will automatically close when the object is deleted.

2.3.1.3. Basic Concepts

The following are basic concepts related to using the Results File Reader.

2.3.1.3.1. Vector Format Description

The Results File Reader offers the ability to get the values of a record of the results file via the CVEC<T> object, where T is the type of scalars of the record of interest (typically int, double, float or complex<double>).

You can create and allocate a CVEC object by typing:

```
CVEC<double> V(10); // Create and allocate a vector of 10 double values
```

The `C_RstFile` object allocates the vector using the correct size.

The size of a vector is accessible by calling the member function:

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
int size = V.Dim();
```

To access the values of the vector, get the pointer to the beginning of the vector:

```
double *values = V.Adr();
```

Or you can directly access the *i*th value:

```
int           i = 0;      // The first value
double    val = V[i];
```

2.3.1.3.2. Get a Record in the Results File

Based on the tree representation shown in Figure 2.1: Results File Structure (p. 81), you can get any record from an opened results file just by specifying its location in the tree.

1. First, the global header of the results file can be accessed using a special call:

   ```
   C_Record      *RstHeader = RstFile.GetGlobalHeader();        // The main global RST File Header (“RST”
   ```

   The results file header contains global information about what can be found in the current file (mesh, results, and so on).

   To get the array of values from a `C_Record` object, use this call:

   ```
   CVEC<int>     VecHead;                        // VecHead is defined as an Vector of Integers.
   RstFile.GetRecord( *RstHeader, VecHead);      // Fills the VecHead vector
   VecHead.Print();                              // Print to screen the values of the vector
   ```

   You can then access the values of this vector. Please refer to the `C_RstFile::E_RstHeader C++ enum` for a list of usable constants.

   ```
   int nnod = VecHead[C_RstFile::E_NNOD];        // To Get the number of Nodes
   int nelm = VecHead[C_RstFile::E_NELM];        // To Get the number of Elements
   ```

2. Next, from this global header you can get the `C_Record` object by giving the complete path to the record of interest. Here we access the nodal solution (NSL) of the first solution set:

   ```
   C_Record *Block = RstHeader->find( "RST::DSI::SET1::NSL");
   ```

3. Then, depending on the type of values the record contains, you can get the values:

   ```
   CVEC<double> V;
   RstFile.GetRecord( *Block, V);
   for( int ii=0; ii<V.Dim(); ii++)          // Print the values of the vector
     cout << ii << ":  " << V[ii] << endl;
   ```

2.3.1.3.3. Repeatedly Reading Records of the Results File

Some sets of records are repeated in the file. To optimize the reading of such records, you can use the dedicated functions `C_RstFile::GetFirstRec` and `C_RstFile::GetNextRec`. They can be used for:

- The element descriptions:

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

`RST::GEO::EID`

- The element type:

  `RST::GEO::ETY`

- The element section data:

  `RST::GEO::SEC`

- The coordinate system description:

  `RST::GEO::CSY`

See Extract the Elements (p. 86) for examples of how to use these functions.

2.3.1.4. Extract Nodes and Elements

From the VecHead global header vector, you can get global information for the nodes.

- Maximum node number of the model:

  `C_RstFile::E_MAXN`

- Actual number of nodes used in the solution phase:

  `C_RstFile::E_NNOD`

- Number of DOFs per node:

  `C_RstFile::E_NUMDOF`

2.3.1.4.1. Extract the Nodes

Node locations are stored in the record "LOC", under the geometry section of the tree:

```
c   LOC     dp     nnod      7      (64 bit version)
c                                   Node,X,Y,Z,THXY,THYZ,THZX for each node
c                                   Nodes are in node number order
```

To get the entire vector of node numbers and coordinates, you can use this direct call:

```
// ===== Node,X,Y,Z,THXY,THYZ,THZX for each node. Nodes are in node number order
CVEC<>    GeoNod;    // Type is not specified, double is the default

Block = RstHeader->find( "RST::GEO::LOC");     // Get a handle to the LOC block in the tree
RstFile.GetRecord( *Block, GeoNod, ALL_REC);   // Fills the GeoNod vector; ALL_REC means ask for all nodes

cout << "\n   Global Nodes Vector, Dim =  : " << GeoNod.Dim();

// ===== The same data, but for a single node :
int           NumNod(1);
CVEC<>        GeoNod_i;

RstFile.GetRecord( *Block, GeoNod_i, NumNod);

cout << "\n   Single Nodes Vector, Dim =  : " << GeoNod_i.Dim() << endl;
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.3.1.4.2. Extract the Elements

From the Global header vector, you can get global information for the elements.

- Maximum element number of the model:

  `C_RstFile::E_MAXE`

- Actual number of elements used in the solution phase:

  `C_RstFile::E_NELM`

Element Description Record

Element descriptions are stored in the record “EID”, in the geometry section of the tree:

```
c   EID      i       1     nelm     Element descriptions index table. This
c                                   record contains the record pointers for each
c                                   element description. (LONGINT (2*nelm) for
c                                   64 bit version, relative to ptrEIDL).
c                                   The order of the elements is the same as
c                                   the order in the element equivalence table.

c   ---      i     nelm  10+nodelm  Element descriptions. Each of these records
c                                   is pointed to by a record pointer given in
c                                   the record labeled EID. The length of these
c                                   records varies for each element (actual
c                                   length is returned from routine BINRD8).
c                                   nodelm shown here is the number of nodes for
c                                   this element. Its value is defined in the
c                                   element type description record.
```

You can, for instance, get repeatedly all the element descriptions by calling the functions `C_RstFile::GetFirstRec` and `C_RstFile::GetNextRec`:

```
CVEC<int> ElemDesc;
int iel = RstFile.GetFirstRec( "RST::GEO::EID", ElemDesc, 0, true);
while (iel != -1)
  {
    iel = RstFile.GetNextRec( ElemDes);
  }
```

Element Type Record

Element type descriptions are stored in the record "ETY":

```
c   ETY      i       1     maxety   The element types index table. This record
c                                   contains record pointers for each element
c                                   type description. (Relative to ptrETYPL
c                                   for 64 bit version)
```

In the same way as shown above for element descriptions, you can loop over the element types, calling the same set of functions.

```
CVEC<int> ElemType;
int iel = RstFile.GetFirstRec( "RST::GEO::ETY", ElemType, 0, true);
while (iel != -1)
  {
    iel = RstFile.GetNextRec( ElemType);
  }
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.3.1.5. Extract a Solution Vector

To get a given nodal or element solution in a given solution set, you can call the `C_RstFile::GetSolution` function.

```
CVEC<T> *V = RstFile.GetSolution(int iset, string IdSol, CVEC<T> *Sol, int NumEntity);
```

where:

iset is the solution set number.

IdSol is the solution record ID (for example, "NSL", "RF", "ENS", and so on).

Sol is the vector where the values are to be stored. If it is not allocated, the function allocates it and returns.

NumEntity is the node or element number. Default = All_REC, meaning all entities in the same vector are requested.

You can access the vector properties using this:

```
int Size = SolNod.Dim();  // To get the size of the output vector
double val = SolNod[ii];  // To access the values of the vector
```

2.3.1.6. Example: Extract a Nodal Solution Vector

This example demonstrates reading the nodal solution corresponding to the first solution set.

```
int              iSet = 1;               // ===== Solution set number
string           IdSol = "NSL";          // ===== Solution Id we want to get
CVEC<double>     Displacement;           // ===== The vector to fill with the solution
int              NumEntity = ALL_REC;    // ===== The Entity (node/elem) number; ALL_REC means all entitites

RstFile.GetSolution( iset, IdSol, &Displacement, NumEntity);

cout << "Size of the nodal solution vector : " << Displacement.Dim() << endl;

Displacement.Print( stdout);
```

2.3.1.7. Example: Extract an Element Solution Vector

This example demonstrates reading the element nodal stresses corresponding to the first solution set and the first element.

```
int              iSet = 1;          // ===== Solution set number
string           IdSol = "ENS";     // ===== Solution Id we want to get
CVEC<double>     EnsVector;         // ===== The vector to fill with the solution
int              NumEntity = 1;     // ===== The element number

RstFile.GetSolution(iset, IdSol, &EnsVector, NumEntity);

cout << "Size of the element solution vector : " << EnsVector.Dim() << endl;

EnsVector.Print( stdout);
```

Given an elementary result, we can also loop over all elements where this result is stored using the C_RstFile::GetFirstLocalSolution and C_RstFile::GetNextLocalSolution functions.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
int              iSet = 1;           // ===== Solution set number
string           IdSol = "ENS";      // ===== Solution Id we want to get
CVEC<double>     EnsVector;          // ===== The vector to fill with the solution
int              NumEntity = 1;      // ===== The first element number

NumEntity = RstFile.
GetFirstLocalSolution( iSet, IdSol, EnsVector, NumEntity);


While ( NumEntity > 0)
{
      // EnsVector contains nodal stresses of the element #NumEntity

      NumEntity
      = RstFile.
      GetNextLocalSolution( EnsVector, NextEntity);
      }
```

2.3.1.8. Example: Using the Results File Reader in a Standalone Program

```
/**
* @file   TestRstReader.cpp
*
* @brief  This source file shows the way we can open an RST File,
*  and extract data using the C++ C_RstFile Object.
*
*/

#include <stdio.h>
#include "C_RstFile.h"

void
main(int argc, char *argv[])
{
 char  *FileName = argv[1];

 // ========================================================================
 // ===== Open the RST File :

 cout << "\n Open the RST File  : " << FileName << endl;

 C_RstFile RstFile(FileName);    // ===== OPEN THE RST FILE

 C_Record *RstHeader = RstFile.GetGlobalHeader();  //       Handle on the Global header

 cout << "\n Ansys Rev. : " << RstFile.AnsRev() << endl;

 // ===== Get global numbers for this model

 int maxn = RstFile.RstHeader(C_RstFile::E_MAXN); // Maximum node number
 int nnod = RstFile.RstHeader(C_RstFile::E_NNOD); // Number of nodes

 int maxe = RstFile.RstHeader(C_RstFile::E_MAXE); // Maximum element number
 int nelm = RstFile.RstHeader(C_RstFile::E_NELM); // Number of elements

 cout << "\n Number of Nodes  : " << nnod;
 cout << "\n Max Node Number  : " << maxn << endl;
 cout << "\n Number of Elements  : " << nelm;
 cout << "\n Max Elt Number  : " << maxe << endl;

 // ========================================================================
 // ===== Extract Nodes Data

 // Node,X,Y,Z,THXY,THYZ,THZX for each node. Nodes are in node number order

 C_Record *Block = NULL;

 Block = RstHeader->find("RST::GEO::LOC");  // Get the pointer to the LOC Record
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

```
CVEC<> GeoNod;
RstFile.GetRecord(*Block, GeoNod, ALL_REC); // Fills the GeoNod Array

cout << "\n Global Nodes Vector, Size = : " << GeoNod.Dim();

// The same data, but for a single node:

int  NumNod(1);
CVEC<> GeoNod_i;
RstFile.GetRecord(*Block, GeoNod_i, NumNod); // We only read the data for the 1st node

cout << "\n Single Nodes Vector, Size = : " << GeoNod_i.Dim() << endl;

// ================================================================================
// ===== Extract Elements Data

// ===== Mapping Elt Index <-> Elt Number

Block = RstHeader->find("RST::ELM");  // Get the pointer to the ELM Record

CVEC<int> Elm;
RstFile.GetRecord(*Block, Elm);  // Fills the Elm Array

cout << "\n Element Equivalence Table Size : " << Elm.Dim() << endl;


// ===== Get the Geometry Header

Block = RstHeader->find("RST::GEO");  // Get the pointer to the GEO Record

CVEC<int> GeoHeader;
RstFile.GetRecord(*Block, GeoHeader);  // Fills the GeoHeader Array

// This GeoHeader contains information such as the maximum element type reference number in the model:

int maxety = GeoHeader[C_RstFile::E_GMAXETY];

// For each element type, we can get the Type description

cout << "\n Extract All Element Types Descriptions .... " << endl;

CVEC<int> ElemEty;
int ityp = RstFile.GetFirstRec("RST::GEO::ETY", ElemEty, 0, true);
while (ityp != -1)
{
 cout << " Got Description for Element Type : " << ElemEty[1] << endl;
 ityp = RstFile.GetNextRec(ElemEty);
}

// For each element, we extract the description associated to

cout << "\n Extract All Element Descriptions .... \n";

CVEC<int> ElemDes;
int iel = RstFile.GetFirstRec("RST::GEO::EID", ElemDes, 0, true);
while (iel != -1)
{
 int imat = ElemDes[0],
  itype = ElemDes[1];

 cout << "Element " << iel << ": Material = " << imat << " , Element Type Number = " << itype << endl;

 iel = RstFile.GetNextRec(ElemDes);
}

cout << "\n              Element " << nelm << "   OK" << endl;

// ================================================================================
// ===== We get the number of solution sets

int  NbSet = RstFile.RstHeader(C_RstFile::E_NSETS);
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
cout << "\n Number of Result Sets : " << NbSet << endl;

// Get the Nodal Solution of the 1st solution set

int   iset(1);  // 1st solution set
string IdSol = "NSL"; // “NSL” means Nodal Solution
CVEC<> SolNod;  // This vector will contain the Nodal Solution

RstFile.GetSolution(iset, IdSol, &SolNod);

cout << "Nodal Solution vector : size = " << SolNod.Dim() << endl;


// Get the Nodal Stresses for the 1st Element, for the 1st solution set

IdSol = "ENS";

int  NumElem = 1;
int  Size(0);
CVEC<> EnsElem;

try {

 RstFile.GetSolution(iset, IdSol, &EnsElem, NumElem);
}
catch (C_CadoeException e) {
 if (e.getExceptionCode() == E_ResultNotAvailable)
 {
  cerr << "This result " << IdSol << " is not available\n";
 }
}

Size = EnsElem.Dim();

cout << "\n Size of the ENS vector for the 1st Element = " << Size << endl;


// Loop to get all the Element Nodal Stresses, for the 1st Solution Set

NumElem = RstFile.GetFirstLocalSolution( iset, IdSol, EnsElem, 1); // 1 means we start at the 1st Element
if (NumElem == -3) cerr << "No Elementary results stored in this Result File\n";
if (NumElem == -2) cerr << "this result is not stored for this Element " << NumElem << "in this Solution Set\n

while(NumElem > 0)
{
 cout << "For Element #" << NumElem << " Norm of the Nodal Stress Vector is : " << EnsElem.Nrm() << endl;

 NumElem = RstFile.GetNextLocalSolution( EnsElem);
}

 return;
}
```

2.3.2. Results File Access Routines

The method of accessing the results file described in this section is presented as an alternative to the Results File Reader (p. 81).

You can use the low-level routines described in Accessing Mechanical APDL Binary Files (p. 69) to retrieve data from the results file. Or you can use the routines described below that retrieve data specific to the format of the results file.

These files reside in the subdirectory `\Program Files\ANSYS Inc\V212\ANSYS\customize\user` (on Windows systems) or `/ansys_inc/v212/ansys/customize/misc` (on Linux

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

systems). See Access Routines for Results Files (p. 69) for information on compiling and linking these routines.

2.3.2.1. Overview of the Routines

For each data record in the results file, routines exist that:

- Read the record index and allocate space for the data. These are named `ResRd`*`record`*`Begin`, where *`record`* is a descriptive name of the record, for example, `ResRdNodeBegin`
- Read the data itself. These are named `ResRd`*`record`*, for example, `ResRdNode`
- Deallocate space for the data. These are named `ResRd`*`record`*`End`, for example, `ResRdNodeEnd`

Below is a complete listing of all the routines with the indentation indicating the required nested calling sequence:

```
function ResRdBegin (Nunit,Lunit,Fname,ncFname,Title,JobName,Units,NumDOF,DOF,UserCode,MaxNode,NumNode,MaxElem,
  subroutine ResRdGeomBegin (MaxType,MaxReal,MaxCsys,nXYZ)
    subroutine ResRdTypeBegin (NumType,TypeLen)
      function ResRdType (itype,ielc,TypeNo)
    subroutine ResRdTypeEnd
    subroutine ResRdRealBegin (NumReal,NumPerReal,RealLen)
      function ResRdReal (iReal,Rcon,RealNo)
    subroutine ResRdRealEnd
    subroutine ResRdCsysBegin (NumCsys,CsysLen)
      function ResRdCsys (iCsys,CsysNo)
    subroutine ResRdCsysEnd
    subroutine ResRdNodeBegin
      function ResRdNode (iNode,xyzang)
    subroutine ResRdNodeEnd
    subroutine ResRdElemBegin
      function ResRdElem (iElem,nodes,ElemData)
    subroutine ResRdElemEnd
  subroutine ResRdGeomEnd
  subroutine ResRdSectMatBegin (MaxSect,MaxMat)
    subroutine ResRdSectBegin (NumSect,NumPerSect,SectLen)
      function ResRdSect (iSect,SecData,SectNo)
    subroutine ResRdSectEnd
    subroutine ResRdMatBegin (NumMat,NumPerMat)
      function ResRdMat (iMat,iprop,MatData)
    subroutine ResRdMatEnd
  subroutine ResRdSectMatEnd
  function ResRdSolBegin (key,lstep,substep,ncumit,kcmplx,time,Title,DofLab)
    subroutine ResRdDispBegin
      function ResRdDisp (node,Disp)
    subroutine ResRdDispEnd
    subroutine ResRdRforBegin (nRForce)
      function ResRdRfor (node,idof,value)
    subroutine ResRdRforEnd
    subroutine ResRdBCBegin (BCHeader)
      subroutine ResRdFixBegin (BCHeader,nFixed)
        function ResRdFix (node,idof,value)
      subroutine ResRdFixEnd
      subroutine ResRdForcBegin (BCHeader,nForces)
        function ResRdForc (node,idof,value)
      subroutine ResRdForcEnd
    subroutine ResRdBCEnd
    subroutine ResRdEresBegin
      function ResRdEstrBegin  (iElem)
        function ResRdEstr (iStr,Str)
      subroutine ResRdEstrEnd
    subroutine ResRdEresEnd
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
  subroutine ResRdSolEnd
subroutine ResRdEnd
```

These routines are contained in the file ResRd.F. See the demonstration routine ResRd-Demo.F (p. 80) on the distribution medium for an example of the usage of these routines.

The memory allocation scheme is described in Memory Management Routines (p. 278) in Part 2: Guide to User-Programmable Features (p. 125).

The following sections describe the data-reading routines. See the file ResRd.F and its corresponding include deck ResRd.inc for listings of the corresponding Begin/End routines.

2.3.2.2. ResRdBegin (Opening the File and Retrieving Global Information)

```
*deck,ResRdBegin
      function ResRdBegin (Nunit, Lunit, Fname, ncFname, Title, JobName,
     x                     Units, NumDOF, DOF, UserCode,
     x                     MaxNode, NumNode, MaxElem, NumElem,
     x                     MaxResultSet,NumResultSet)
c primary function:    Open result file and return global information

c object/library: ResRd

c  input arguments:
c     Nunit    (int,sc,in)       - Fortran Unit number for file (ANSYS uses 12)
c     Lunit    (int,sc,in)       - Current print output unit (usually 6 <STDOUT>)
c     Fname    (ch*(ncFname),sc,in) - The name (with extension) for the file
c     ncFname  (int,sc,in)       - Number of characters in Fname

c  output arguments:
c     Title    (ch*80,ar(2),out) - Title and First subtitle
c     JobName  (ch*32,sc,out)    - Jobname from file
c     Units    (int,sc,out)      - unit system
c                                    = 0 - user defined units
c                                    = 1 - SI
c                                    = 2 - CSG
c                                    = 3 - U.S. Customary, using feet
c                                    = 4 - U.S. Customary, using inches
c                                    = 5 - MKS
c                                    = 6 - MPA
c                                    = 7 - uMKS
c     NumDOF   (int,sc,out)      - Number of DOF per node
c     DOF      (int,ar(*),out)   - The DOFs per node
c     UserCode (int,sc,out)      - Code for this application
c     MaxNode  (int,sc,out)      - Maximum node number used
c     NumNode  (int,sc,out)      - Number of nodes attached to elements
c     MaxElem  (int,sc,out)      - Maximum element number used
c     NumElem  (int,sc,out)      - Number of elements used
c     MaxResultSet (int,sc,out)  - Maximum number of result sets (usually 1000)
c     NumResultSet (int,sc,out)  - Number of result sets on file
c     ResRdBegin (int,sc,out)    - 0, successful  other, error in file open
```

2.3.2.3. ResRdGeomBegin (Retrieving Global Geometry Information)

```
*deck,ResRdGeomBegin
      subroutine ResRdGeomBegin (MaxType, MaxReal, MaxCsys, nXYZ)
c primary function:    Read Geometry Header Record

c object/library: ResRd
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c  input arguments:  none

c  output arguments:
c     MaxType  (int,sc,out)       - Maximum element type
c     MaxReal  (int,sc,out)       - Maximum real constant set number
c     MaxCsys  (int,sc,out)       - Maximum coordinate system number
c     nXYZ     (int,sc,out)       - number of nodes with coordinates
```

2.3.2.4. ResRdType (Retrieving Element Types)

```
*deck,ResRdType
      function ResRdType (itype,ielc,TypeNo)
c primary function:    Read an element type record

c object/library: ResRd

c  input arguments:
c     itype    (int,sc,in)         - Element type number

c  output arguments:  none
c     ielc     (int,ar(IELCSZ),out)- Element characteristics
c     TypeNo    (int,sc,out)       - External Element Type number
c     ResRdType (int,sc,out        - number of words read
```

2.3.2.5. ResRdReal (Retrieving Real Constants)

```
*deck,ResRdReal
      function ResRdReal (iReal,Rcon,RealNo)
c primary function:    Read real constant record

c object/library: ResRd

c  input arguments:
c     iReal    (int,sc,in)             - Real set number

c  output arguments:  none
c     Rcon      (dp,ar(ResRdReal),out) - Real Constants
c     RealNo    (int,sc,out)           - External Real Constant number
c     ResRdReal (int,sc,out)           - Number of real constants in set
```

2.3.2.6. ResRdCsys (Retrieving Coordinate Systems)

```
*deck,ResRdCsys
      function ResRdCsys (iCsys,Csys,CsysNo)
c primary function:    Read a coordinate system record

c object/library: ResRd

c  input arguments:
c     iCsys     (int,sc,in)           - Coordinate system number

c  output arguments:
c     Csys      (dp,ar(ResRdCsys),out)- Coordinate system description
c     CsysNo    (int,sc,out)          - External Coordinate system number
c     ResRdCsys (int,sc,out)          - Number of values
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c  output arguments:
```

2.3.2.7. ResRdNode (Retrieving Nodal Coordinates)

```
*deck,ResRdNode
      function ResRdNode (iNode,xyzang)
c primary function:    Get a node

c object/library: ResRd

c  input arguments:
c     iNode     (int,sc,in)     - node sequence number
c                                  (1 to nXYZnode)

c  output arguments:
c     xyzang    (dp,ar(6),out)  - x,y,z,thxy,thyz,thzx for node
c     ResRdNode (int,sc,out)    - Node number
```

2.3.2.8. ResRdElem (Retrieving Elements)

```
*deck,ResRdElem
      function ResRdElem (iElem, nodes, ElemData)
c primary function:    Read an element

c object/library: ResRd

c  input arguments:
c     iElem    (int,sc,in)       - The element number

c  output arguments:
c     ResRdElem(int,sc,out)      - Number of nodes
c     nodes    (int,ar(n),out)   - Element nodes
c     ElemData (int,ar(10),out)  - Element information
c                                    mat    - material reference number
c                                    type   - element type number
c                                    real   - real constant reference number
c                                    secnum - section number
c                                    esys   - element coordinate system
c                                    death  - death flag
c                                             = 0 - alive
c                                             = 1 - dead
c                                    solidm - solid model reference
c                                    shape  - coded shape key
c                                    elnum  - element number
c                                    pexcl  - P-Method exclude key
```

2.3.2.9. ResRdSectMatBegin (Retrieving Global Section and Material Information)

```
*deck,ResRdSectMatBegin
      subroutine ResRdSectMatBegin (MaxSect, MaxMat)
c primary function:    Read maximum section and material number
c                      from the Geometry Header Record

c object/library: ResRd
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c  input arguments:  none

c  output arguments:
c     MaxSect  (int,sc,out)       - Maximum section number
c     MaxMat   (int,sc,out)       - Maximum material number
```

2.3.2.10. ResRdSect (Retrieving Section Data)

```
*deck,ResRdSect
      function ResRdSect (iSect,SecData,SectNo)
c primary function:    Read section record

c object/library: ResRd

c  input arguments:
c     iSect    (int,sc,in)             - Section set number

c  output arguments:
c     SecData   (dp,ar(ResRdSect),out) - Section data
c     SectNo    (int,sc,out)           - Exteral Section number
c     ResRdSect (int,sc,out)           - Number of section data in set
```

2.3.2.11. ResRdMat (Retrieving Material Data)

```
*deck,ResRdMat
      function ResRdMat (iMat,iprop,MatData)
c primary function:    Read material record

c object/library: ResRd

c  input arguments:
c     iMat     (int,sc,in)             - Material set number
c     iprop    (int,sc,in)             - Property reference number
c                                        See mpinqr for details

c  output arguments:
c     MatData   (dp,ar(ResRdMat),out)  - Material data for type iprop
c     ResRdMat  (int,sc,out)           - Number of material data in set
```

See Function mpinqr (Getting Information About a Material Property) (p. 300) for details on the property reference number (iprop).

2.3.2.12. ResRdSolBegin (Retrieving Result Set Location)

```
*deck,ResRdSolBegin
      function ResRdSolBegin (key,lstep,substep,ncumit,kcmplx,time,
     x                        Title,DofLab)
c primary function:    Read the solution header records

c object/library: ResRd

c  input arguments:
c     key      (int,sc,in)       - 0, find by set number
c                                  1, find by lstep/substep
c                                  2, find by ncumit
```

```
c                                  3, find by time
c     lstep    (int,sc,in/out)   - Load step number
c                                   if key=0, this is the set number
c     substep  (int,sc,in/out)   - Substep of this load step
c     ncumit   (int,sc,in/out)   - Cumulative iteration number
c     kcmplx   (int,sc,in)       - 0, Real solution   1, Imaginary solution
c     time     (dp,sc,in/out)    - Current solution time

c  output arguments:
c     Title    (ch*80,ar(5),out)  - Title and 4 subtitles
c     DofLab   (ch*4,ar(nDOF),out)- Labels for DOFs
c     ResRdSolBegin (int,sc,out)  - 0, requested solution set found
c                                   1, not found
```

2.3.2.13. ResRdDisp (Retrieving Nodal Solution)

```
*deck,ResRdDisp
      function ResRdDisp (node,Disp)
c primary function:    Retrieve a nodal displacement

c object/library: ResRd

c  input arguments:
c     node     (int,sc,in)       - Node number

c  output arguments:  none
c     Disp     (dp,ar(nDOF),out) - Displacements
c     ResRdDisp(int,sc,out)      - Number of displacements
```

2.3.2.14. ResRdRfor (Retrieving Reaction Solution)

```
*deck,ResRdRfor
      function ResRdRfor (node,idof,value)
c primary function:    Retrieve a reaction force

c object/library: ResRd

c  input arguments:
c     node     (int,sc,in)       - External node number
c     idof     (int,sc,in)       - Internal dof number

c  output arguments:
c     value     (dp,sc,in)       - Value of reaction force
c     ResRdRfor (int,sc,out)     - Number of returned values (0 or 1)
```

2.3.2.15. ResRdFix (Retrieving Applied Nodal Constraints)

```
*deck,ResRdFix
      function ResRdFix (node,idof,value)
c primary function:    Retrieve a constraint value

c object/library: ResRd

c  input arguments:
c     node     (int,sc,in)       - External node number
c     idof     (int,sc,in)       - Internal dof number
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c  output arguments:
c     value    (dp,ar(4),in)     - Real,Imag, RealOld,ImagOld
c     ResRdFix (int,sc,out)      - Number of returned values (0 or 4)
```

2.3.2.16. ResRdForc (Retrieving Applied Nodal Loads Solution)

```
*deck,ResRdForc
      function ResRdForc (node,idof,value)
c primary function:    Retrieve an applied force value

c object/library: ResRd

c  input arguments:
c     node      (int,sc,in)       - External node number
c     idof      (int,sc,in)       - Internal dof number

c  output arguments:
c     value     (dp,ar(4),in)     - Real,Imag, RealOld,ImagOld
c     ResRdForc (int,sc,out)      - Number of returned values (0 or 4)
```

2.3.2.17. ResRdEstr (Retrieving Element Solutions)

```
*deck,ResRdEstr
      function ResRdEstr (iStr,Str)
c primary function:   Get an element's results

c object/library: ResRd


c  input arguments:
c     iStr    (int,sc,in)       - element record number (1-NUMELEDATASETS)

c  output arguments:
c     ResRdEstr (int,sc,out)       - Number of element values
c     Str       (dp,ar(nStr),out)  - element values
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 3: The CDWRITE (CDB) File Format

This chapter discusses how to write a coded database file, `Jobname.CDB`, that can be used to export a model from Mechanical APDL into another application. The `Jobname.cdb` file contains model data in terms of Mechanical APDL input commands.

The following topics are discussed:

3.1. Using the CDWRITE Command

To export a model from Mechanical APDL to another application, use menu path **Main Menu> Preprocessor> Archive Model> Write** or the **CDWRITE** command within the general preprocessor, PREP7. This produces a coded database file called `Jobname.cdb`. You specify the jobname using **Utility Menu> File> Change Jobname** or the **/FILNAME** command. If you supply no jobname, Mechanical APDL uses the default name "file".

The `Jobname.cdb` file contains selected geometry (nodes and elements), load items, and other model data in terms of Mechanical APDL input commands. (For a complete list of data in the file, see the **CDWRITE** description in the *Command Reference*.) You can convert this information to a format compatible with the program into which you are importing it. The next few pages describe special considerations and commands you may need to do this conversion.

Note:

Files created by the **CDWRITE** command have the active coordinate system set to Cartesian (**CSYS**, 0).

Mechanical APDL may create parameters in the **CDWRITE** file that start with an underscore (_), usually an "_z." Such parameters are for Mechanical APDL internal use and pass information to the Mechanical APDL GUI.

3.1.1. Customizing Degree of Freedom Labels: the /DFLAB Command

Mechanical APDL uses a set of default labels for the degrees of freedom. You use these labels when entering boundary conditions, or Mechanical APDL uses the labels when writing the `Jobname.cdb` file.

You can change the labels to reflect the degrees of freedom of the other program by issuing the command **/DFLAB**. If you are customizing the degree-of-freedom labels, **/DFLAB** must be the first command you enter within Mechanical APDL. You may want to include the command in your `START.ANS` file. You can use **/DFLAB** only at the Begin processing level.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

/DFLAB assigns or reassigns the "displacement" and "force" labels in the Mechanical APDL degree-of-freedom list. For example, DOF number 1 is predefined to have a displacement label of UX and a force label of FX, but you can assign new labels to this degree of freedom by issuing **/DFLAB**. Changing predefined labels generates a warning message.

The format for the **/DFLAB** command is:

```
/DFLAB,NDOF,LabD,LabF
```

NDOF

Mechanical APDL degree-of-freedom number (1 to 32)

LabD

Displacement degree of freedom label to be assigned (up to four characters)

LabF

Force label to be assigned (up to four characters)

You can also use **/DFLAB** to assign labels to spare degree-of-freedom numbers. Spare displacement and force labels are from 27 to 32. All other degree-of-freedom numbers are predefined, as follows:

DOF Number	Corresponding Displacement Label	Corresponding Force Label
1	UX	FX
2	UY	FY
3	UZ	FZ
4	ROTX	MX
5	ROTY	MY
6	ROTZ	MZ
7	AX	CSGX
8	AY	CSGY
9	AZ	CSGZ
10	VX	VFX
11	VY	VFY
12	VZ	VFZ
13	GFV1	GFL1
14	GFV2	GFL2
15	GFV3	PFFL
16	WARP	BMOM
17	CONC	RATE
18	HDSP	DVOL
19	PRES	FLOW
20	TEMP	HEAT

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DOF Number	Corresponding Displacement Label	Corresponding Force Label
21	VOLT	AMPS
22	MAG	FLUX
23	ENKE	NPKE
24	ENDS	NPDS
25	EMF	CURT
26	CURR	VLTG

3.2. Coded Database File Commands

In the coded database file `Jobname.CDB`, most Mechanical APDL commands have the same format they have elsewhere. (See the *Command Reference* for command-specific information.) However, the format for some commands differs slightly in the `Jobname.CDB` file. The format for these commands is described below. Commands that use an unblocked format include the label UNBL in one of the command fields.

The **CDWRITE** command has an UNBLOCKED and a BLOCKED option. The UNBLOCKED option writes all data out in command format. The default BLOCKED option writes certain data items in a fixed format, including those that could potentially contain large amounts of data, such as nodal data.

3.2.1. BFBLOCK Command

BFBLOCK defines a block of nodal body-force loads. This is the recommended method for inputting nodal body-force loads into the Mechanical APDL database. The command syntax is:

```
BFBLOCK,NUMFIELD,Lab,NDMAX,NDSEL,TAB,–,MESHFLAG
Format
```

NUMFIELD

The number of fields in the blocked format.

Lab

The body-force load label used to describe the block data. (For a list of load labels, see **BF**.)

NDMAX

The maximum node number defined.

NDSEL

The number of selected nodes.

–

Reserved.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MESHFLAG

When using nonlinear adaptivity in a linear analysis (**NLGEOM**,OFF), specifies how to apply nodal body-force loading on the mesh. Valid only when *Lab* = HGEN or TEMP.

0 – Nodal body-force loading occurs on the current mesh (default).

1 – Nodal body-force loading occurs on the initial mesh for nonlinear adaptivity.

Format

Data descriptors defining the format.

The format of the body-force load block is as follows:

- Field 1 - Node number.
- Fields 2-7 - Body-force load values to apply to the node. The number of fields is dependent on the body-force load label. (See **BF**.)

The final line of the block format is always a **BF** command with -1 for the node number.

The following example shows a typical **BFBLOCK** formatted set of body-force loads (using non-tabular input) that define a temperature load (TEMP).

```
BFBLOCK,2,TEMP,        97,        97,0
(i9,6(pg16.9))
        1      300.000000
        2      300.000000
        3      300.000000
 .
 .
 .
       95      300.000000
       96      300.000000
       97      300.000000
BF,end,LOC,       -1,
```

For an example using tabular input, see the **BFEBLOCK** command (p. 102).

In the GUI, the **BFBLOCK** command must be contained in an externally prepared file and read into Mechanical APDL (via **CDREAD**, **/INPUT**, or other commands).

The **BFBLOCK** command is not valid in a ***DO** loop.

3.2.2. BFEBLOCK Command

BFEBLOCK defines a block of element body-force loads. This is the recommended method for inputting element body-force loads into the Mechanical APDL database. The command syntax is:

```
BFEBLOCK,NUMFIELD,Lab,ELMAX,NUMLOADS,TAB
Format
```

NUMFIELD

The number of fields in the blocked format.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Lab

The body-force load label used to describe the block data. (See **BFE** for a list of load labels.)

ELMAX

The maximum element number defined.

NUMLOADS

The number of body loads written.

TAB

Key for tabular input:

0 – Non-tabular input.

1 – Tabular input.

Format

Data descriptors defining the format.

The format of the body-force load block is as follows:

- Field 1 - Element number.
- Field 2 - Starting location for entering data.
- Fields 3 - Body load value for the specified starting location. (See **BFE** for information about starting location and associated body-force load values.)

The final line of the block format is always a **BFE** command with -1 for the element number.

The following example shows a typical **BFEBLOCK** formatted set of body-force loads (using tabular input) that define a temperature load (TEMP).

```
BFEBLOCK,3,TEMP,       108,       108,1
(2i9,a)
         1        1         %BODYFORCE%
         2        1         %BODYFORCE%
         3        1         %BODYFORCE%
.
.
.
       106        1         %BODYFORCE%
       107        1         %BODYFORCE%
       108        1         %BODYFORCE%
BFE,end,LOC,       -1,
```

The following example shows a typical **BFEBLOCK** formatted set of body-force loads (using non-tabular input) that define a force load (FORC) with complex input.

```
BFEBLOCK,3,FORC,       80,       480,0
(2i9,1(pg16.9))
         1        1 -27.5000000
         1        2  37.5000000
         1        3  47.5000000
         1        4 -22.5000000
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
         1        5  12.5000000
         1        6  17.5000000
.
.
.
        80        1 -27.5000000
        80        2  37.5000000
        80        3  47.5000000
        80        4 -22.5000000
        80        5  12.5000000
        80        6  17.5000000
BFE,end,LOC,       -1,
```

In the GUI, the **BFEBLOCK** command must be contained in an externally prepared file and read into Mechanical APDL (via **CDREAD**, **/INPUT**, or other commands).

The **BFEBLOCK** command is not valid in a ***DO** loop.

3.2.3. CE Command

CE defines the constant term in a constraint equation. The command format in `Jobname.CDB` is:

```
CE,UNBL,Type,LENGTH,NCE,CONST
```

Type

The type of data to be defined. DEFI is the valid label.

LENGTH

The total number of variable terms in the constraint equation.

NCE

The constraint equation reference number.

CONST

The constant term of the equation.

Another version of the **CE** command defines the variable terms in a constraint equation. You must issue this version of the command after the **CE** command described above. This command repeats until all terms are defined.

The alternate format for the **CE** command is:

```
CE,UNBL,Type,N1,Dlab1,C1,N2,Dlab2,C2
```

Type

The type of data to be defined. NODE is the valid label.

N1

The node number of the next term.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Dlab1

The degree-of-freedom label of *N1*.

C1

The coefficient of *N1*.

N2

The node number of the next term.

Dlab2

The degree-of-freedom label of *N2*.

C2

The coefficient of *N2*.

3.2.4. CP Command

CP defines a coupled-node set. You repeat the command until all nodes are defined. The command format in Jobname.CDB is:

```
CP,UNBL,LENGTH,NCP,Dlab,N1,N2,N3,N4,N5,N6,N7
```

LENGTH

The total number of nodes in the coupled set

NCP

The coupled node reference number

Dlab

The degree of freedom label for the set

N1,N2,N3,N4,N5,N6,N7

The next seven node numbers in the coupled set

3.2.5. CMBLOCK Command

CMBLOCK defines the entities contained in a node or element component. The command format in Jobname.CDB is:

```
CMBLOCK,Cname,Entity,NUMITEMS,,,,,KOPT
Format
```

Cname

Eight character component name.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Entity

Label identifying the type of component (NODE or ELEMENT).

NUMITEMS

Number of items written.

--

Reserved for future use.

--

Reserved for future use.

--

Reserved for future use.

--

Reserved for future use.

KOPT

Controls how element component contents are updated during nonlinear mesh adaptivity analysis:

0 – Component is not updated during remeshing and therefore contains only initial mesh elements (default).

1 – Component is updated during remeshing to contain the updated elements.

This argument is valid only for nonlinear mesh adaptivity analysis with *Entity* = ELEM, and for solid element components only. This argument does not support NLAD-ETCHG analysis.

Format

Data descriptors defining the format. For **CMBLOCK** this is always (8i10).

The items contained in this component are written at 10 items per line. Additional lines are repeated as needed until all *NumItems* are defined. If one of the items is less than zero, then the entities from the item previous to this one (inclusive) are part of the component.

The **CMBLOCK** command is not valid in a ***DO** loop.

3.2.6. CYCLIC Command

CYCLIC,CDWR defines the input and output of a cyclic symmetry analysis. The syntax is:

```
CYCLIC,CDWR,Value1,Value2,Value3,...
```

The following describes the values written to the .CDB file for cyclic options **CYCLIC**,CDWR:

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

***Value1* = 1**

Value2

Number of cyclic sectors

Value3

Number of solutions in cyclic space

Value4

Harmonic index of this load

Value5

Cyclic coordinate system

Value6

< 0 or Static: only solve for given harmonic indices from **CYCOPT**,HIND

> 0: tolerance for the Fourier load

***Value1* = 2**

Value2

Cyclic edge type (0 = undefined; 1 = areas; 10 = lines; 100 = keypoints; 1000 = nodes)

Value3

0 or blank

Value4

Maximum possible harmonic index

Value5

Force load coordinate system (1 = global coordinate system; 0 = cyclic coordinate system)

Value6

Inertia load coordinate system (1 = global coordinate system; 0 = cyclic coordinate system)

***Value1* = 3 - 22**

Value2* - *Value6

Cyclic edge constraint equation/coupling degree of freedom (0 = all available degrees of freedom; otherwise bitmap) for pair IDs 1-5

(Repeat as necessary for other pair IDs (*Value1* = 4 - 22))

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

***Value1* = 23 - 30**

Cyclic harmonic index bit bins (each bin holds 32 harmonic indices by 5 containers corresponding to *Value2* - *Value6*)

Value2* - *Value6

Cyclic harmonic index bits (0 = solve for harmonic index; nonzero values indicate skipped harmonic indices)

***Value1* = 31**

Value2

Max node number in base sector

Value3

Max element number in base sector

Value4

Number of defined nodes in base sector

Value5

Number of defined elements in base sector

***Value1* = 32**

Value2

/CYCEXPAND number of sectors to expand (total)

Value3

Number of edge component pairs

Value4* - *Value8

/CYCEXPAND number of sectors to expand (per window)

***Value1* = 33**

Not used

***Value1* = 34**

Cyclic **CSYS** coordinate system integer data (part 1)

Value2

Theta singularity key

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Value3

Phi singularity key

Value4

Coordinate system type

***Value1* = 35**

Cyclic **CSYS** coordinate system integer data (part 2)

Value2

Coordinate system number

Value3

Not used (defaults to 0)

Value4

Not used (defaults to 0)

***Value1* = 36**

Value2

Number of user-defined cyclic edge pair components

Value3

Rotate cyclic edge nodes into cyclic coordinate system (0 = rotate edge nodes (default); 1 = do not rotate edge nodes)

Value4

NLGEOM flag (0 = no **NLGEOM** effects (default); 1 = include **NLGEOM** effects)

Value5

Sector edge display key (-1 = suppresses display of edges between sectors even if the cyclic count varies between active windows; 0 = averages stresses or strains across sector boundaries. This value is the default (although the default reverts to 1 or ON if the cyclic count varies between active windows); 1 = no averaging of stresses or strains occurs and sector boundaries are shown on the plot)

***Value1* = 101**

Value2

Sector angle (degrees)

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Value3

XYZ tolerance input for matching low/high nodes

Value4

Angle tolerance for matching low/high nodes (degrees)

Value5

Tolerance in the element coordinate system for unequal meshes

***Value1* = 102 - 104**

Cyclic **CSYS** coordinate system double precision data (part 1)

Value2* - *Value4

Coordinate system transformation matrix (total of 9 values)

***Value1* = 105**

Cyclic **CSYS** coordinate system double precision data (part 2)

Value2* - *Value4

Origin location (XYZ)

***Value1* = 106**

Cyclic **CSYS** coordinate system double precision data (part 3)

Value2

Used for elliptical, spheroidal, or toroidal systems. If CSYS = 1 or 2, *Value2* is the ratio of the ellipse Y-axis radius to X-axis radius (defaults to 1.0 (circle))

Value3

Used for spheroidal systems. If CSYS = 2, *Value3* is the ratio of ellipse Z-axis radius to X-axis radius (defaults to 1.0 (circle))

***Value1* = 107**

Cyclic **CSYS** coordinate system double precision data (part 4)

Value2

First rotation about local Z (positive X toward Y)

Value3

Second rotation about local X (positive Y toward Z)

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Value4

Third rotation about local Y (positive Z toward X)

Value1 **= 121**

Value2

Root of component names defining low and high ranges

Value1 **= 122**

Value2

Cyclic low/high xref array parameter name (node)

Value1 **= 123**

Value2

Cyclic low/high xref array parameter name (line)

Value1 **= 124**

Value2

Cyclic low/high xref array parameter name (area)

Value1 **= 125**

Value2

The component name of the elements to expand (see **/CYCEXPAND**,,WHAT)

Value1 **= 201**

Value2

Total number of modes extracted during a cyclic modal solve. This value is only available after call to **CYCCALC**.

Value3

Mode superposition flag to limit results written to `.MODE` and `.RST` files

Value4

Excitation engine order

Value1 **= 202**

Value2

Type of mistuning (1 = stiffness; 2 = mass; 3 = both; -1 = use user macro CYCMSUPUSERSOLVE)

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Value3

Cyclic mode superposition restart flag (1 = new frequency sweep; 2 = new mistuning parameters; -1 = form blade superelement and stop)

Value4

Cyclic mode superposition key to perform complex modal analysis of reduced system

Value5

Number of CMS modes for mistuned reduced order model (see **CYCFREQ**,BLADE)

***Value1* = 203**

Value2

Array name for aerodynamic coupling coefficients

***Value1* = 204**

Unused

***Value1* = 205**

Value2

The name of the nodal component containing the blade boundary nodes at the blade-to-disk interface (see **CYCFREQ**,BLADE). This is used for cyclic mode superposition analyses that include mistuning or aero coupling.

***Value1* = 206**

Value2

The name of the element component containing the blade superelements (see **CYCFREQ**,BLADE). This is used for cyclic mode superposition analyses that include mistuning or aero coupling.

***Value1* = 207**

Value2

The name of the array holding stiffness mistuning parameters

***Value1* = 208**

Unused

***Value1* = 209**

Rotational velocity from the base linear perturbation analysis.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Value2

X-component of rotational velocity

Value3

Y-component of rotational velocity

Value4

Z-component of rotational velocity

***Value1* = 210**

Value2

Beginning of frequency range for CMS modes (see **CYCFREQ**,BLADE). This is used for cyclic mode superposition analyses that include mistuning or aero coupling.

Value3

End of frequency range for CMS modes (see **CYCFREQ**,BLADE). This is used for cyclic mode superposition analyses that include mistuning or aero coupling.

***Value1* = 211**

Value2

Number of modes for a cyclic mode superposition damped modal solve

Value3

Beginning of frequency range for cyclic mode superposition damped modal solve

Value4

End of frequency range for cyclic mode superposition damped modal solve

3.2.7. EBLOCK Command

EBLOCK defines a block of elements. The command syntax is:

```
EBLOCK,NUM_NODES,Solkey,ELMAX,ELSEL
Format
```

NUM_NODES

The number of nodes to be read in the first line of an element definition.

Solkey

The solid model key. The element is part of a solid model if the keyword SOLID appears here. When *Solkey* = SOLID, Field 8 (the element shape flag) may be left at zero, and Field 9 is the number of nodes defining this element.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ELMAX

The maximum element number defined.

ELSEL

The number of selected elements.

Format

Data descriptors defining the format.

The format of the element block is as follows for the SOLID format:

- Field 1 - The material number.
- Field 2 - The element type number.
- Field 3 - The real constant number.
- Field 4 - The section ID attribute (beam section) number.
- Field 5 - The element coordinate system number.
- Field 6 - The birth/death flag.
- Field 7 - The solid model reference number.
- Field 8 - The element shape flag.
- Field 9 - The number of nodes defining this element if *Solkey* = SOLID; otherwise, Field 9 = 0.
- Field 10 - *Not used.*
- Field 11 - The element number.
- Fields 12-19 - The node numbers. The next line will have the additional node numbers if there are more than eight.

The format without the SOLID keyword is:

- Field 1 - The element number.
- Field 2 - The type of section ID.
- Field 3 - The real constant number.
- Field 4 - The material number.
- Field 5 - The element coordinate system number.
- Fields 6-15 - The node numbers. The next line will have the additional node numbers if there are more than ten.

The final line of the block will be a -1 in field 1.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you are in the GUI, the **EBLOCK** command must be contained in an externally prepared file and read into Mechanical APDL (via **CDREAD**, **/INPUT**, or other commands).

The **EBLOCK** command is not valid in a ***DO** loop.

3.2.8. EN Command

EN is used to define an element . If an element contains more than eight nodes, the **EN** command is repeated until all nodes are defined. The command format in `Jobname.CDB` is:

```
EN,UNBL,Type,NUMN,I1,I2,I3,I4,I5,I6,I7,I8
```

Type

The type of data to be defined. Valid labels are ATTR (read in element attributes), and NODE (read in nodes defining the element).

NUMN

The number of nodes.

I1,I2,I3,I4I5,I6,I7,I8

The integer values to be read:

- If *Type* is ATTR, the integer values are the element attributes. Attributes are in the order: NUMN,MAT,TYPE,REAL,SECNUM,ESYS,NUMELEM,SOLID,DEATH,EXCLUDE
- If *Type* is NODE, the integer values are the node numbers.

3.2.9. LOCAL Command

LOCAL defines a local coordinate system. The command format in `Jobname.CDB` is:

```
LOCAL,UNBL,Type,NCSY,CSYTYP,VAL1,VAL2,VAL3
```

Type

The type of data to be defined. Valid labels are LOC (read in system origin), ANG (read in rotation angles), and PRM (read in system parameters).

NCSY

The coordinate system reference number.

CSYTYP

The coordinate system type (0, 1, 2, or 3).

VAL1,VAL2,VAL3

Values to be read:

- If *Type* is LOC, values are the system origin in global Cartesian coordinates.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If *Type* is ANG, values are the rotation angles in degrees.
- If *Type* is PRM, values are the first and second parameters of the system.

3.2.10. M Command

M defines a master degree of freedom. The command format in `Jobname.CDB` is:

```
M,UNBL,NODE,Dlab
```

NODE

The node number

Dlab

The degree-of-freedom label

3.2.11. MPDATA Command

MPDATA defines a material property data table. You repeat the command until all properties are defined. The command format in `Jobname.CDB` is:

```
MPDATA,UNBL,LENGTH,Lab,MAT,STLOC,VAL1,VAL2,VAL3
```

LENGTH

The total number of temperatures in the table.

Lab

The material property label. (For valid labels, see **MP**.)

MAT

The material reference number.

STLOC

The starting location in the table for the next three property values.

VAL1,VAL2,VAL3

Property values assigned to three locations in the table starting at *STLOC*.

3.2.12. MPTEMP Command

MPTEMP defines a temperature table. You repeat the command until all temperature values are defined. The command format in `Jobname.CDB` is:

```
MPTEMP,UNBL,LENGTH,STLOC,TEMP1,TEMP2,TEMP3
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

LENGTH

The total number of temperatures in the table

STLOC

The starting location in the table for the next three temperature values

TEMP1,TEMP2,TEMP3

Temperatures assigned to three locations in the table starting at *STLOC*

3.2.13. N Command

If the UNBLOCKED option is used with **CDWRITE**, then **N** defines a node. The command format in Jobname.CDB is:

```
N,UNBL,Type,NODE,SOLID,PARM,VAL1,VAL2,VAL3
```

Type

The type of data to be defined. Valid labels are LOC (read in coordinates) and ANG (read in rotation angles).

NODE

The node number.

SOLID

The solid model reference key. Not present for *Type*= ANG.

PARM

Line parameter value (0 if not on line). Not present for *Type*= ANG.

VAL1,VAL2,VAL3

Values to be read:

- If *Type* is LOC, values are the coordinates in the global Cartesian system.
- If *Type* is ANG, values are the rotation angles in degrees.

3.2.14. NBLOCK Command

NBLOCK defines a block of nodes. This is the recommended method for inputting nodes into the Mechanical APDL database. The command syntax is:

```
NBLOCK, NUMFIELD, Solkey, NDMAX, NDSEL
Format
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

NUMFIELD

The number of fields in the blocked format.

Solkey

The solid model key. The node is part of a solid model if the keyword SOLID appears here.

NDMAX

The maximum node defined.

NDSEL

The number of nodes written.

Format

Data descriptors defining the format.

The format of the node block is as follows:

- Field 1 - Node number.
- Field 2 - The solid model entity (if any) in which the node exists (if SOLID key).
- Field 3 - The line location (if the node exists on a line and if SOLID key).
- Field 4 - 6 - The nodal coordinates.
- Field 7 - 9 - The rotation angles (if *NUMFIELD* > 3).

Only the last nonzero coordinate/rotation is output; any trailing zero values are left blank.

The final line of the block is always an **N** command using a -1 for the node number.

The following example shows a typical **NBLOCK** formatted set of node information. Note that this example has no rotational data. It contains only the first six fields.

```
NBLOCK,6,SOLID,       849,       723
(3i9,6e21.13e3)
        1        0        0 8.7423930292124E-001 7.1843141243360E-001 8.2435547360131E-001
        3        0        0 9.2314873336026E-001 9.3459943382943E-001 4.8406643591666E-001
        4        0        0 1.1410427242574E+000 7.6883495387624E-001 2.5867801436812E-001
 .
 .
 .
      847        0        0 6.2146469267794E-001 8.0122597436764E-001 8.1352232529497E-001
      848        0        0 8.1179373384170E-001 6.6711479947438E-001 7.6547291135454E-001
      849        0        0 7.4952223718564E-001 7.6089019544242E-001 7.4112247735703E-001
N,UNBL,LOC,       -1,
```

If you are in the GUI, the **NBLOCK** command must be contained in an externally prepared file and read into Mechanical APDL (**CDREAD**, **/INPUT**, or other commands).

The **NBLOCK** command is not valid in a ***DO** loop.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.2.15. *PREAD Command

```
*PREAD,Par,NUMVALS
Format
END PREAD
```

Par

Alphanumeric name to identify this parameter.

NUMVALS

Number of values.

Format

Data descriptor defining the format of the subsequent lines (as needed). The format is always (4g20.13).

3.2.16. R Command

R defines a real constant set. You repeat the command until all real constants for this set are defined. The command format in Jobname.CDB is:

```
R,UNBL,NSET,Type,STLOC,VAL1,VAL2,VAL3
```

NSET

The real constant set reference number.

Type

The type of data to be defined. LOC is the valid label.

STLOC

The starting location in the table for the next three constants.

VAL1,VAL2, VAL3

Real constant values assigned to three locations in the table starting at *STLOC.*

3.2.17. RLBLOCK Command

RLBLOCK defines a real constant set. The real constant sets follow each set, starting with Format1 and followed by one or more Format2's, as needed. The command format is:

```
RLBLOCK,NUMSETS,MAXSET,MAXITEMS,NPERLINE
Format1
Format2
```

NUMSETS

The number of real constant sets defined

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MAXSET

Maximum real constant set number

MAXITEMS

Maximum number of reals in any one set

NPERLINE

Number of reals defined on a line

Format1

Data descriptor defining the format of the first line. For **RLBLOCK**, this value is always (2i8,6g16.9). The first i8 is the set number, the second i8 is the number of values in this set, followed by up to six real constant values.

Format2

Data descriptors defining the format of the subsequent lines (as needed); this is always (7g16.9).

The real constant sets follow, with each set starting with Format1, and followed by one or more Format2's as needed.

RLBLOCK is not valid in a ***DO** loop.

3.2.18. SE Command

SE defines a superelement. The command format in `Jobname.CDB` is:

```
SE,UNBL,File,,,TOLER,TYPE,ELEM
```

File

The name (case-sensitive) of the file containing the original superelement matrix created by the generation pass (*Sename*.SUB).

TOLER

Tolerance for determining whether use-pass nodes are noncoincident with master nodes having the same node numbers. Default = 0.0001.

TYPE

Element type number.

ELEM

Element number.

This command command also appears in the *Command Reference*. The format shown here contains information specific to the **CDREAD**/**CDWRITE** file.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.2.19. SECBLOCK Command

SECBLOCK for Beams

SECBLOCK retrieves all mesh data for a user-defined beam section as a block of data. You repeat the command for each beam section that you want to read. The command format is:

```
SECBLOCK
Format1
Format2
Format3
```

Format1

The First Line section. The first value is the number of nodes, and the second is the number of cells.

Format2

The Cells Section. The first 9 values are the cell connectivity nodes. The 10th (last) value is the material ID (**MAT**).

Format3

The Nodes Section. This section contains as many lines as there are nodes. In this example, there are 27 nodes, so a total of 27 lines would appear in this section. Each node line contains the node's boundary flag, the Y coordinate of the node, and the Z coordinate of the node. Currently, all node boundary flags appear as 0s in a cell mesh file. Because all node boundary flags are 0, **SECBLOCK** ignores them when it reads a cell mesh file.

Sample User Section Cell Mesh File

Following is a sample excerpt from a custom section mesh file for a section with 27 nodes, 4 cells, and 9 nodes per cell:

First Line:

```
27   4
```

Cells Section:

```
      1   3   11   9    2   6    10   4    5    2
      7   9   23   21   8   16   22   14   15   1
      9   11  25   23   10  18   24   16   17   1
      11  13  27   25   12  20   26   18   19   1
```

Nodes Section:

```
...
0     0.0          0.0
0     0.025        0.0
0     0.05         0.0
0     5.0175       0.0
0     19.98        10.00
0     20.00        10.00
...
```

SECBLOCK for Shells

SECBLOCK can retrieve data for shell sections. The command format is:

```
SECBLOCK,N
TKn,MATn,THETAn,NUMPTn
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

N

Total number of layers. The second line (*TKn*, *MATn*, *THETAn*, *NUMPTn*) is repeated *N* times (once for each layer).

TKn

Layer thickness for layer number *n*

MATn

Material ID for layer number *n* (defaults to element material ID)

THETAn

Layer orientation angle for layer number *n*

NUMPTn

Number of integration points in layer number *n*

SECBLOCK is not valid in a ***DO** loop.

3.2.20. SFBEAM Command

SFBEAM defines a surface load on selected beam elements. Remaining values associated with this specification are on a new input line with a 4(1pg16.9) format. The command format in `Jobname.CDB` is:

```
SFBEAM,ELEM,LKEY,Lab,UNBL,DIOFFST,DJOFFST
```

ELEM

The element number.

LKEY

The load key associated with these surface loads.

Lab

A label indicating the type of surface load. PRES (for pressure) is the only valid label.

DIOFFST

Offset distance from node I.

DJOFFST

Offset distance from node J.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.2.21. SFE Command

If the UNBLOCKED option is used with **CDWRITE**, then **SFE** defines a surface load. Values associated with this specification are on a new input line with a 4(1pg16.9) format. The command format in Jobname.CDB is:

```
SFE,ELEM,LKEY,Lab,KEY,UNBL
```

ELEM

The element number.

LKEY

The load key associated with this surface load.

Lab

A label indicating the type of surface load. (For a list of load labels, see **SFE**.)

KEY

A value key. The possible values and meaning of each value depend on the specified load label. (See the *KVAL* argument of **SFE** for a list of value keys based on load label.)

3.2.22. SFEBLOCK Command

SFEBLOCK defines a block of element surface loads. This is the recommended method for inputting element surface loads into the Mechanical APDL database. The command syntax is:

```
SFEBLOCK,NUMFIELD,Lab,ELMAX,NUMLOADS,TAB
Format
```

NUMFIELD

The number of fields in the blocked format.

Lab

The surface load label used to describe the block data. (For a list of load lables, see **SFE**.)

ELMAX

The maximum element number defined.

NUMLOADS

The number of surface loads written.

TAB

Key for tabular input:

0 – Non-tabular input.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1 – Tabular input.

Format

Data descriptors defining the format.

The format of the surface load block is as follows:

- Field 1 - Element number.
- Fields 2 - Load key or face number associated with the surface load.
- Fields 3 - Value key. (See the *KVAL* argument of **SFE** for a list of value keys based on load label.)
- Fields 4-7 - Surface load values to apply to each element.

The final line of the block format is always an **SFE** command with -1 for the element number.

The following example shows a typical **SFEBLOCK** formatted set of body-force loads (using non-tabular input) that define a convection load (CONV) characterized by film coefficient and bulk temperature.

```
SFEBLOCK,4,CONV,         5,        24,0
(i9,i4,i4,6(pg16.9))
        3   1   1  10.0000000      10.0000000      0.00000000      0.00000000
        3   1   2  300.000000      300.000000      0.00000000      0.00000000
        4   1   1  6.50000000      6.50000000      0.00000000      0.00000000
        4   1   2  146.153800      146.153800      0.00000000      0.00000000
        5   1   1  3.50000000      3.50000000      0.00000000      0.00000000
        5   1   2  300.000000      300.000000      0.00000000      0.00000000
SFE,end,LOC,       -1,
```

In the GUI, **SFEBLOCK** must be contained in an externally prepared file and read into Mechanical APDL (via **CDREAD**, **/INPUT**, or other commands).

SFEBLOCK is not valid in a ***DO** loop.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Part 2: Guide to User-Programmable Features

Chapter 1: Understanding User Programmable Features (UPFs)

The Mechanical APDL program has an open architecture, allowing you to write your own routines or subroutines in C, C++, or Fortran and either link them to the program or use them as external commands. Some standard program features originated as user customizations, also known as *user programmable features (UPFs)*.

You can take advantage of user customization if you are licensed for the Ansys Mechanical Enterprise family of products (Ansys Mechanical Enterprise, Ansys Mechanical Enterprise PrepPost, and Ansys Mechanical Enterprise Solver).

Other Ansys, Inc. products do not support customization of the Mechanical APDL program.

For more information about compilers, see the Ansys, Inc. installation guide specific to your operating system:

Ansys, Inc. Linux Installation Guide

Ansys, Inc. Windows Installation Guide

The following customization topics are available:

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1.1. What Are User Programmable Features?

User programmable features are tools you can use to write your own routines. Using UPFs, you can tailor the Mechanical APDL program to your organization's needs. For instance, you may need to define a new material behavior, a special element, a contact interfacial model, or a modified failure criterion for composites.

UPFs provide the following capabilities:

- To read information into or retrieve information from the Mechanical APDL database, you can create subroutines and either link them into the program or use them in the external command feature (see Appendix A: Creating External Commands in Linux (p. 383) for more information about external commands). If you link these subroutines into Mechanical APDL, you are limited to 10 database access commands. Such commands, created through either method, operate at all levels of program operation, including the begin, preprocessor, general postprocessor, time-history postprocessor, and solution levels. For more information about accessing the database, see Accessing the Mechanical APDL Database (p. 289).
- Some UPF subroutines enable you to specify various types of loads, including **BF** or **BFE** loads, pressures, convections, heat fluxes, and charge densities. These routines are described under Subroutines for Customizing Loads (p. 252).
- Some UPF subroutines enable you to modify and monitor existing elements. For details, see Subroutines for Modifying and Monitoring Existing Elements (p. 199).
- Some UPF subroutines enable you to define the following material properties: plasticity, creep, swelling law, viscoplasticity, hyperelasticity, and layered element failure criteria. To see inputs and outputs for these routines, see Subroutines for Customizing Material Behavior (p. 204).
- For analyses involving contact, another set of UPF subroutines enables you to define contact properties, friction models, and interaction behaviors. To see inputs and outputs for these routines, see Subroutines for Customizing Contact Interfacial Behavior (p. 237).
- Several sets of UPFs enable you to define new elements and to adjust the nodal orientation matrix. See Creating a New Element (p. 151) for more information.
- You can call Mechanical APDL as a subroutine in a program you have written. To learn how, see Running Mechanical APDL as a Subroutine (p. 266).

1.2. What You Should Know Before Using UPFs

Before you do anything with linked UPFs, contact your on-site system support person to get the permissions needed to access the appropriate Ansys, Inc. files.

The UPF subroutines are written in Fortran; some extensions are used. They contain comments intended to give you enough detail to develop your own versions of the subroutines.

User routines that can be modified have the term "USERDISTRIB" in the first line of the routine. These routines are provided with the Ansys, Inc. distribution media. You can modify these routines to tailor the program to your specific needs. Certain other routines described in this document are not provided on the distribution media, but can be called using the provided header information.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To use UPFs successfully, you need strong working knowledge of the following:

- The Mechanical APDL program.
- The UPF subroutines themselves. Study the UPF subroutines before customizing them, and make sure that you fully understand the subroutines, as well as any applicable functions. Unless you review them carefully, a few UPF subroutines may seem like a maze with many logic paths to consider. You may have to set special variables correctly in order to run your customized Mechanical APDL solution without errors. Even if you have in-depth knowledge of the Mechanical APDL input and your desired outputs, you still need to ensure that everything that must be done in the UPF subroutines is done properly in your custom version.
- Fortran. Besides knowing how to write Fortran subroutines, you must be sure that the level of the Fortran compiler is as least as high as the level mentioned in your Ansys, Inc. installation manual. For more information on Fortran compilers, refer to the installation guide specific to your operating system (Ansys, Inc. Linux Installation Guide or Ansys, Inc. Windows Installation Guide). You also need to know what to do should the computer abort the program due to an arithmetic error, a file read error, a memory access error, and so on.
- The mathematics of the phenomenon you are planning to include.

Important

- UPFs are not available or will behave unpredictably in certain data center environments or on some hardware configurations. You should take special care when using UPFs on parallel systems. It is a good practice to verify your coding with single processing by using the `-np,1` option before you run your analysis. For additional information, consult your installation guide or your on-site Ansys system-support person
- Carefully consider whether you wish to use UPFs, especially if you are linking them into Mechanical APDL (rather than into a shared library for use as external commands). When you add your own routines to Mechanical APDL by either method, you are creating a customized, site-dependent version of the program. Ansys, Inc. considers the use of UPFs a nonstandard use of the program, one that the Ansys, Inc. Quality Assurance verification testing program does not cover. Therefore, you are responsible for verifying that the results produced are accurate and that your customizations do not adversely affect unchanged areas of the Mechanical APDL program.
- Although the flexibility that UPFs offer can be highly attractive, UPF usage is a complicated process that can introduce errors. Consider what you want your customizations to accomplish. You may be able to customize Mechanical APDL more easily and safely with macros than with UPFs.
- When using shared memory, all user-programmable features are supported except for common block variables in Fortran and external variables in C or C++. You should avoid overwriting the values of these variables by multiple cores at the same time.
- For Distributed Ansys, all user-programmable features are supported except for global (often in common block) variables in Fortran and global (often external) variables in C or C++. You should avoid overwriting the values of these variables; they should have the same values across all cores used in the distributed solution.

For other guidelines for nonstandard uses of the Mechanical APDL program, see User-Programmable Features and Nonstandard Uses in the *Advanced Analysis Guide*.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1.3. Planning Your UPFs

UPFs can range from a simple element output routine for customized output to a complex user optimization. Before you start programming, ask yourself these questions:

- Does the capability you want already exist in Mechanical APDL? Remember, a capability may not be obvious at first, especially to a novice user.
- Does your proposed subroutine fit into the Mechanical APDL program architecture and specifications? For example, you can not program a user element that has more than 32 degrees of freedom per node.

Use your experience and judgment to answer these questions. If you need help to do so, consult your ANSYS Support Distributor. If you can respond "no" to the first question and "yes" to the second question, then the user routine you are planning will be both useful and feasible.

1.4. Studying the Mechanical APDL User Routines

Your Ansys, Inc. distribution medium contains the source codes for all user routines:

- If you are running Mechanical APDL under Linux, the source code for the UPF routines resides in directory `/ansys_inc/v212/ansys/customize/user/`.
- If you are running Mechanical APDL under Windows, the source code for the UPF routines resides in directory `Program Files\ANSYS Inc\V212\ansys\customize\user\`.

Most of the user routines have at least simple functionality, so print out the routines of interest before you start programming. All source routines are concatenated onto file `user.f` or `user.for`. Delete the routines you do not want and make appropriate changes to the others.

1.5. Programming in Languages Other Than Fortran

If you wish to run Mechanical APDL with user customizations, the preferred method is to design and program your custom routine in Fortran. Although you can use languages other than Fortran, in each case Fortran must provide the interface to the rest of the Mechanical APDL program. If you do use a language other than Fortran, such as the C or C++, your code may require a Fortran shell.

You need to take care when calling Fortran subroutines from C or C++ subroutines. You must use the symbol associated with the Fortran subroutine when invoking the subroutine from a C or C++ function. This symbol typically differs slightly from the Fortran subroutine name, and is extremely system dependent.

On many Linux systems, you build this symbol name by taking the Fortran subroutine name, converting it to lower case, and appending an underscore. For example, the symbol name for the Fortran subroutine **HeapInquire** would be **heapinquire_**. You would have to use the symbol **heapinquire_** in the invoking C function to avoid an unsatisfied external reference when the program is linked.

Keep in mind that the instance described above is just an example. Compilers from different vendors may construct the symbols differently. Consult the manuals for your specific compiler for information on how to call Fortran subroutines from C or C++ functions.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For more information on Fortran compilers, refer to the installation guide specific to your operating system (Ansys, Inc. Linux Installation Guide or Ansys, Inc. Windows Installation Guide).

Using Python

As an alternative to compiled languages like C and Fortran, you can use the Python language to code user programmable subroutines. You must install a Python distribution before using this feature. A subset of the documented UPF subroutines support the Python UPF capability. For more information, see Using Python to Code UPF Subroutines (p. 371).

1.6. Developing UPFs: a Suggested Strategy

When developing customizations for Mechanical APDL, you can avoid problems and reduce debugging time by following a gradual, orderly process. Start with a trivial test. Then, add a few changes at a time so that if something goes wrong, the error that caused the problem should be isolated and relatively easy to locate.

The example procedure below illustrates this type of gradual process. The example assumes that you are creating a new element using the method described in Creating a New Element by Directly Accessing the Program Database (p. 171). You develop and test it by performing these steps:

1. Get the applicable element subroutines for `uel101` from the product distribution medium. Add a small change (such as a misspelling in an output heading), then compile and link the subroutines.
2. Using a production version of the program, run several analysis problems using various elements to form a base for comparison.
3. Run the same problem using USER101 on your custom version of the program.
4. Compare the results from Steps 2 and 3. If they show discrepancies other than the misspelled output heading, resolve them before you go on to Step 5.
5. Choose the standard Mechanical APDL element that most closely resembles your new custom element, and run some problems on a production version of Mechanical APDL using that element.
6. Modify the element subroutines to match the element you chose in Step 5. Then, compile and link those subroutines into a custom version of Mechanical APDL.
7. Again, compare the results from Steps 5 and 6. If they don't match, resolve the discrepancies before moving on to Step 8.
8. Modify your element subroutines to include the features you want. Then, compile and link the subroutines into a custom version of Mechanical APDL.
9. Test the changes with a series of increasingly complex problems for which you already know the answers.

1.7. Include Decks

In addition to the subroutines and functions described in this chapter, most of the include decks (files with the extension `.inc`) used by Mechanical APDL are on your product distribution medium. The include

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

decks, also called *commons*, contain important but relatively small amounts of data. The program also handles large amounts of data using various access routines (GET and PUT), described elsewhere in this document.

Note:

When you compile a user-programmable feature (UPF) in a shared library (ANSUSER-SHARED.BAT), you cannot access the common block variables. (All such variables will return a value of zero.)

To insert include decks in a subroutine or function, use the INCLUDE (or an analogous) statement. *Do not modify an include deck under any circumstances*. The following table lists some of the more commonly used include files and the definitions they contain:

Include File	Description
acel-cm.inc	Contains accelerations and angular velocities
ansys-def.inc	Defines general Mechanical APDL parameters. You must include this common to retrieve the parameter values of *MEM_INTEGER*, *MEM_DOUBLE*, *MEM_COMPLEX*, or *MEM_REAL*.
cmopt.inc	Contains optimization variables
ech-prm.inc	Defines parameters for element characteristics
el-ccmt.inc	Defines element characteristics (comments only)
ele-com.inc	Contains element-specific information
el-parm.inc	Defines pointers for the element data array
eluc-om.inc	Defines the element degree of freedom pointers
ety-com.inc	Element type data
imp-com.inc	Used by all routines and functions in the program
outp-cm.inc	Defines output control information
soptcm.inc	Contains solution options and keys
stack.inc	Defines stack storage. You must include this common in any routines that access stack space.
step-cm.inc	Contains load step information
us-vrcm.inc	Defines storage of user-defined variables

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1.8. Choosing a Linking Method

After you make your changes to the user routines supplied on your product distribution medium, you can either:

- Link your routines into shared libraries (as discussed starting in Appendix A: Creating External Commands in Linux (p. 383)), or
- Compile and link your custom routines into the Mechanical APDL program itself. This is discussed for Linux systems in Compiling and Linking UPFs on Linux Systems (p. 133) and for Windows systems in Compiling and Linking UPFs on Windows Systems (p. 139). You may need superuser or root privileges to run the procedure that does the linking.

For both Windows and Linux platforms, three methods of compiling and linking are available:

- **/UPF** command
- ANSUSERSHARED script (creates a shared library on Linux or a dynamic-link library on Windows)
- **ANS_ADMIN212** Utility

The **/UPF** command method is typically used by individuals wanting to occasionally use their customized code for certain runs. The advantages of this method are that it is very easy to use and the source code can be displayed in the output file.

The shared library (Linux) and dynamic link library (Windows) methods are typically used to run Mechanical APDL with frequently used user-libraries or third-party libraries (material libraries, and so on). This method is advantageous if customized code is frequently used or shared with other users.

The **ANS_ADMIN212** method is useful for someone wanting to create a permanently changed Mechanical APDL executable which will be used by many users, or used most of the time. Companies that validate their user-customized code might want to consider this option.

In some cases, you might want to combine two of the methods of compiling and linking. The program allows you to combine the **ANS_ADMIN212** method with either the **/UPF** command method or the ANSUSERSHARED method. Note that the **/UPF** command method cannot be combined with the ANSUSERSHARED method.

As an example of combining these methods, you might first create a custom executable with **ANS_ADMIN212** that contains user creep laws. Then, you might use the **ANS_USER_PATH** environment variable to include a user material (or third-party library) created with the ANSUSERSHARED method.

For detailed compiling and linking procedures, see Compiling and Linking UPFs on Linux Systems (p. 133) and Compiling and Linking UPFs on Windows Systems (p. 139).

1.9. Compiling and Linking UPFs on Linux Systems

There are three methods that you can use to link your custom routines into Mechanical APDL:

- Using the /UPF Command (p. 134)
- Creating a Shared Library (p. 135)

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Using the ANS_ADMIN Utility (p. 135)

The source files for the user routines reside in the following subdirectory: `/ansys_inc/v212/ansys/customize/user/`

For all three methods, you can write your user routines in one language or a combination of languages (Fortran, C, and C++). Note that when using a combination of languages, you are responsible for the calling interfaces between languages (see Programming in Languages Other Than Fortran (p. 130)).

The Ansys, Inc. Linux Installation Guide lists the compilers that are required for specific platforms.

Note:

You will need all the compilers specified in the Installation Guide specific to your operating system to use these user programmable features, including GNU GCC 8.2.0. This specific compiler can be downloaded from the Ansys, Inc. customer site. See Downloading and Installing the GCC Compiler (p. 136) for details.

1.9.1. Using the /UPF Command

The **/UPF** command offers the simplest method for linking user programmable features into Mechanical APDL. The format of the command is:

```
/UPF, RoutineName
```

where *RoutineName* is the name of a user routine (`filename.ext`) that you want to link. The specified routine must reside in the current working directory.

To use this method start Mechanical APDL in batch mode and include one or more **/UPF** commands in the specified input listing. When the program reads the input and detects **/UPF**, Mechanical APDL will be relinked automatically.

You can include **/UPF** anywhere in your input file, and you can repeat **/UPF** as many times as needed to include multiple user routines in the relinked version. Any user routine can be linked using this method.

When you run a user-linked version of the program by this method, the output includes the following:

```
NOTE – This Mechanical APDL version was linked by /UPF with n user supplied routine(s).
```

where *n* indicates the number of routines specified by **/UPF** commands. The routines that have been linked will be included in the output listing.

Example Using Mixed Languages

The following directory contains an example of using the **/UPF** command method to link user routines that are written in mixed languages (Fortran, C, C++):

`/ansys_inc/v212/ansys/custom/user/<platform>/Examples`

The `README.txt` file in this directory contains complete instructions on running this example. This is a simple, automated test that verifies whether compilers are correctly installed and configured.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1.9.2. Creating a Shared Library

You can also set up UPFs on some Linux systems through a shared library. All Fortran files (files ending with .F), C files (files ending with .c), and C++ files (files ending in .cpp) that you want to include in your shared library should reside in your working directory. To compile all *.F, *.c, and *.cpp routines, issue the following command:

```
/ansys_inc/v212/ansys/customize/user/ANSUSERSHARED
```

If the compile was successful, you will be asked if a shared file is to be created. Choose **Yes** to create a shared library named libansuser.so.

To use this library, set the **ANS_USER_PATH** environment variable to point to the working directory where the libansuser shared library resides. Use one of the following commands, depending on the Linux shell you are using:

```
setenv ANS_USER_PATH workingdirectory
```

or

```
export ANS_USER_PATH=workingdirectory
```

When you run a user-linked version of Mechanical APDL, the output echos the value of **ANS_USER_PATH** and will include the following:

```
NOTE:  This Mechanical APDL version was linked by Licensee
```

To return to the original version of Ansys, unset the **ANS_USER_PATH** environment variable.

You can use another environment variable, **ANS_USER_PATH_212**, to point to a shared library specific to Release 2021 R2. **ANS_USER_PATH_212** supersedes **ANS_USER_PATH**. This allows you to set up and use more than one shared library containing UPFs. To change back to the shared library specified by **ANS_USER_PATH**, simply unset the **ANS_USER_PATH_212** environment variable.

Ansys, Inc. recommends using the ANSUSERSHARED script as a template to try compilers that are not supported by Ansys, Inc., such as the GNU compilers. To do so, edit the ANSUSERSHARED script, making changes to the appropriate platform logic. Note that if you do use compilers other than those listed in the *ANSYS Installation and Configuration Guide* specific to your operating system, you will need to debug (that is, find missing libraries, unsatisfied externals, etc.) them yourself. Ansys, Inc. does not provide assistance for customers using unsupported compilers or if the resulting objects are not compatible with the executable(s) as distributed.

1.9.3. Using the ANS_ADMIN Utility

As mentioned previously, the source files for the user routines reside in subdirectory /ansys_inc/v212/ansys/customize/user/. If you modify any of these subroutines, you can select the **Relink ANSYS** option from **ANS_ADMIN212** utility to link these changes. This method creates a custom Mechanical APDL executable.

The **Relink ANSYS** option compiles all Fortran files (files ending with .F), C files (files ending with .c), and C++ files (files ending in .cpp) in the current working directory. The procedure then loads all object files (files ending with .o) along with the default Mechanical APDL objects and libraries in

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

`/ansys_inc/v212/ansys/customize/user/<platform>`, where *<platform>* is replaced by the folder representative of your operating system. The new executable file created will be named `ansyscust.e` and will reside in the working directory.

When you run a user-linked version of Mechanical APDL, the output includes the following:

```
NOTE:  This Mechanical APDL version was linked by Licensee
```

If you intend to run the linked version of Mechanical APDL in a distributed environment (for example, on a cluster), the executable (`ansyscust.e`) must reside in the same directory path on all systems. However, you need to link it on only one system; you can then copy the executable to the other systems.

1.9.3.1. Special Considerations for SUSE 15 SP2

For SUSE Linux Enterprise Server 15 SP2, the system-required GCC should be used for linking custom executables via **ANS_ADMIN212**. For access to the system GCC compiler, add the location to your PATH or set **ANS_GCC_PATH** as shown below.

The standard location for the system GCC is `/usr/bin`.

```
setenv PATH /usr/bin:$PATH (for csh/tcsh shell)
export PATH=/usr/bin:$PATH (for sh/bash shell)
```

or

```
setenv ANS_GCC_PATH /usr/bin (for csh/tcsh shell)
export ANS_GCC_PATH=/usr/bin (for sh/bash shell)
```

If GCC is set to a version other than the system GCC via PATH or **ANS_GCC_PATH**, you may see this error:

```
/usr/lib64/libpthread_nonshared.a(pthread_atfork.oS): unable to initialize decompress status for section .debug
/usr/lib64/libpthread_nonshared.a: error adding symbols: File format not recognized
```

1.9.4. Downloading and Installing the GCC Compiler

In order to link UPFs on a Linux system, the Ansys, Inc. built version of the GCC 8.2.0 compiler is required. Therefore, you must download this compiler from the Ansys, Inc. customer site and extract the necessary files using these steps:

1. From the Ansys customer site, www.ansys.com/customercommunity, click **Downloads** > **Current Release**.
2. Select the **Linux x64** operating system.
3. Select **Primary Packages** from the **Select Download Type** drop-down menu.
4. Expand the **Tools** menu.
5. Click the **GCC Compiler** download option.
6. Select your desired download directory and click **Save**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This will download the compressed zip file `linux-toolchain-8.2.0-01.zip`.

7. Create a `linux-toolchain-8.2.0-01` folder to contain the contents of the zip file:

```
mkdir linux-toolchain-8.2.0-01
```

8. Extract the file within the location created in step 7.

```
unzip linux-toolchain-8.2.0-01.zip
```

9. Add the location to your PATH, or set **ANS_GCC_PATH** for access to the GCC 8.2.0 compiler. For example:

```
setenv PATH <installed_location>/linux-toolchain-8.2.0-01/bin:$PATH (for csh/tcsh shell)
export PATH=<installed_location>/linux-toolchain-8.2.0-01/bin:$PATH (for sh/bash shell)
```

or

```
setenv ANS_GCC_PATH <installed_location>/linux-toolchain-8.2.0-01/bin (for csh/tcsh shell)
export ANS_GCC_PATH=<installed_location>/linux-toolchain-8.2.0-01/bin (for sh/bash shell)
```

The GCC version should display gcc (ansys-20190911) 8.2.0.

1.10. Sharing Data Between User Routines

Mechanical APDL allows more than one user routine in a single run. You can issue multiple **/UPF** commands in the input file to activate the routines.

Using multiple routines simultaneously may require sharing data generated by one routine with another. The easiest method for doing so is to use common-block (or global) variables.

In Linux, a single shared library contains all compiled user routines and data sharing is straightforward. In Windows, however, each user routine is built into a separate dynamic link library (DLL); to share data, functions and data must be explicitly exported and imported.

The userdata (p. 265) subroutine enables you to store the common-block functionality and data. You can edit usercm (p. 266) (included in `userdata`) to add common-block variables. The compiler uses **!DEC$ ATTRIBUTES DLLEXPORT** and **!DEC$ ATTRIBUTES DLLIMPORT** to indicate which functions and common-block variables to export or import, respectively. Both commands are valid for the supported Intel Fortran and C compilers.

Example 1.1: Creating and Exporting Functions and Common Block Variables

```
c usercm.inc
*comdeck,usercm USERDISTRIB
!DEC$ ATTRIBUTES DLLEXPORT :: /usercm/
        common /usercm/  userdatsz,userdatptr
        pointer (userdatptr,userdataarray)
        integer userdatsz
  double precision userdataarray(*)
c
c userdat.F . Sample function exported for use in other subroutines
      function getusercmvals(iloc,sz,outdata)
!DEC$ ATTRIBUTES DLLEXPORT :: getusercmvals
#include "usercm.inc"
      integer         iloc,sz,getusercmvals,iX
      double precision outdata(*)
```

```
   double precision userdataloc(*)
      pointer (userdatptr,userdataloc)
      if (iloc.lt.1.or.iloc+sz-1.gt.userdatsz) then
          getusercmvals = 0
      else
    do iX=iloc,iloc+sz-1,1
   outdata(iX)= userdataloc(iX)
    enddo
          getusercmvals = 1
      endif
      return
      end
```

Example 1.2: Importing and Using Functions

Imported functions are added to the interface section. Common blocks are also imported as needed.

```
        INTERFACE
        FUNCTION getusercmvalsz ()
!DEC$ ATTRIBUTES DLLIMPORT :: getusercmvalsz
        integer getusercmvalsz
        END
        return
        end
        END INTERFACE

!DEC$ ATTRIBUTES DLLIMPORT :: /usercm/
        common /usercm/  userdatsz,userdatptr
        pointer (userdatptr,userdataarray)
        integer userdatsz
        double precision userdataarray(*)
```

Using common-block variables in shared memory requires care. Multiple threads started by the executable access and share the same memory location and can overwrite each other's values. To minimize conflict, allocate enough space for each thread and avoid writing to the same location at the same time.

Example 1.3: Safely Allocating Separate Data for Each Thread

One double-precision location exists in the common block. Enough space is allocated for *numberofprocessors* * 1 locations. Functions `pplock` and `ppunlock` are used when initializing the memory or values.

When setting the value, write to the iy memory location only (used for that specific thread). Even if other threads access the same common block, they do not modify that memory location.

```
c -----------------------
c Initialize common-block values in user routine 1 such as USolBeg.F
c
  call pplock(LOCKUPF)
  inumprocs = ppinqr(PP_NPROCS)
  call initusercmvals(inumprocs)
     val1(:) = 0.0d0
  iy = setusercmvals(ix, inumprocs,val1)
  call ppunlock(LOCKUPF)
c
c -----------------------
c Set common-block value to time in user routine 2
c
  isize = 1
c Go to location allocated for the iy the thread
  iy = ppproc()+1
  val1 = time
c Set just the iy-th value.
  iy = setusercmvals(iy,isize,val1)
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c
c ----------------------
c Get saved value from common block in user routine 3
c
  iproc = ppproc()+1
  isize = 1
  call getusercmvals(iproc,isize,dFldTime)
```

For information about the pplock (p. 353), ppinqr (p. 352), ppunlock (p. 353), and ppproc (p. 353) subroutines used in Example 1.3: Safely Allocating Separate Data for Each Thread (p. 138), see Subroutines for Your Convenience (p. 351).

1.11. Compiling and Linking UPFs on Windows Systems

There are three methods that you can use to link your custom routines into the Mechanical APDL program:

Use the **/UPF** command (p. 140)

Create a dynamic-link library (p. 143)

Use the ANS_ADMIN Utility (p. 144)

The source files for the user routines reside in the following subdirectory: `Program Files\ANSYS Inc\V212\ansys\customize\user\`.

The user programmable features are loaded onto the system only if you perform a custom installation and choose to install the customization tools. If you intend to modify any of the user routines, make a duplicate copy of the `Program Files\ANSYS Inc\V212\ansys\customize\user\` directory to preserve the original files for later use, if necessary.

For all three methods, you can write your user routines in one language or a combination of languages (Fortran, C, and C++). Note that when using a combination of languages, you are responsible for the calling interfaces between languages (see Programming in Languages Other Than Fortran (p. 130)).

The Ansys, Inc. Windows Installation Guide lists the compilers that are required for Windows systems.

Note:

You will need all the compilers specified in the Installation Guide specific to your operating system to use these user programmable features. Ansys, Inc. does not provide assistance if customers are using unsupported compilers, or if the resulting objects are not compatible with the Mechanical APDL executable(s) as distributed.

Before linking UPFs, make sure that the INCLUDE, LIB, and PATH environment variables are set correctly for the required C/C++ and Intel Fortran compilers.

Visual Studio 2017 Professional is also required for linking user programmable features on Windows platforms. In Visual Studio 2017, C++ is not installed by default. To install C++, you must select **Custom Install** and select **Common Tools for C++ for 2017**.

Before using any of the methods described below for linking UPFs, open the following Command Prompt window if you have Visual Studio 2017 Professional installed:

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Start > All apps > Visual Studio 2017 > Developer Command Prompt for VS2017

Note:

For all three linking methods, you might have issues with write/modify access if you use the default working directory: `Program Files\ANSYS Inc\V212\ansys\custom\user\`. If the linking operation errors out or fails, try running the required applications as an administrator by right-clicking and choosing "Run as administrator" while launching each application.

For all three linking methods, you can set the **ANS_USER_PATH** environment variable to specify the working directory where the created UPF DLLs reside. You can use another environment variable, **ANS_USER_PATH_212**, to point to a set of UPF DLLs specific to Release 2021 R2. **ANS_USER_PATH_212** supersedes **ANS_USER_PATH**. This allows you to set up and use more than one set of UPF DLLs. To change back to the location specified by **ANS_USER_PATH**, simply unset the **ANS_USER_PATH_212** environment variable.

1.11.1. Using the /UPF Command

The **/UPF** command offers the simplest method for linking user programmable features into Mechanical APDL. The format of the command is:

```
/UPF, RoutineName
```

where *RoutineName* is the name of the user routine (`filename.ext`) that you want to link. The specified routine must reside in the current working directory.

Following are prerequisites for using the **/UPF** command method on a Windows system:

- A script named `findUPF.bat` is used to detect the **/UPF** command. You must include the path to this script in your system PATH variable. This script is typically located in `Program Files\Ansys Inc\V212\ansys\bin\<platform>` where <*platform*> is a directory that uniquely identifies the hardware platform version: "Winx64" for 64-bit Windows.
- Before starting Mechanical APDL, you must set the **ANS_USE_UPF** environment variable to TRUE. This causes the program to search for **/UPF** in the input file. This environment variable is required only on Windows systems and only when using the **/UPF** command method for linking UPFs. The **ANS_USE_UPF** environment variable should not be set when using other methods to link UPFs.

To use this method start Mechanical APDL in batch mode and include one or more **/UPF** commands in the specified input listing. When the program reads the input and detects **/UPF**, the appropriate DLL is created.

You can include **/UPF** anywhere in your input file, and you can repeat **/UPF** as many times as needed to include multiple user routines in the relinked version. The following user routines can be linked using this method:

UANBEG

UANFIN

UCNVRG

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

UELMATX
UITBEG
UITFIN
ULDBEG
ULDFIN
USER01 -USER10
USERCNPROP
USERCREEP
USERCV
USERCZM
USERELEM
USERFLD
USERFRIC
USERFX
USERHYPER
USERINTER
USERMAT
USERMATTH
USEROU
USERSWSTRAIN
USER_TBELASTIC
USERWEAR
USOLBEG
USOLFIN
USRCAL
USREFL
USRSHIFT
USRSURF116
USSBEG
USSFIN
UTIMEINC

If you are running in a distributed memory parallel environment (Distributed Ansys), you must set the ANS_USER_PATH environment variable to the working directory and include the name of the head compute node:

```
set ANS_USER_PATH=\\headnode\workingdirectory
```

When you run a user-linked version of the program by this method, the output includes this message:

```
NOTE – This ANSYS version was linked by /UPF with n user supplied routine(s).
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where *n* indicates the number of routines specified by **/UPF** commands. The routines that have been linked will be included in the output listing.

Example Using Mixed Languages

The following directory contains an example of using the **/UPF** command method to link user routines that are written in mixed languages (Fortran, C, C++):

`Program Files\ANSYS Inc\V212\ansys\custom\user\<platform>\Examples`

The `README.txt` file in this directory contains complete instructions on running this example. This is a simple, automated test that verifies whether compilers are correctly installed and configured.

1.11.1.1. Using the /UPF Command on a Windows HPC Server System

Running a distributed memory parallel (Distributed Ansys) solution on a Windows HPC Server system is more involved than running across a simple Windows cluster. You can use the steps described here to test the **/UPF** command on a Windows HPC Server system. Several files are included with the Ansys, Inc. software installation for use in this example.

Before you begin, Visual Studio 2017 Professional and Intel Fortran must be on the head compute node.

The files needed for this example can be found in the following directory:

`C:\Program Files\ANSYS Inc\v212\commonfiles\MPI\WindowsHPC\UPF`

These include two user routines, a Mechanical APDL input file, and two files required by the HPC Job Manager:

```
usercreep.F
usermat.F
slupf.inp
RUNANSYS_UPF.xml
runansysupf.bat
```

Copy these files to:

`C:\Temp\%USERNAME%`

You will need to modify the value of the **ANS_USER_PATH** environment variable in `RUNANSYS_UPF.xml` to the appropriate location where the UPF DLL library resides.

Submit `RUNANSYS_UPF.xml` to the HPC Job Manager. When the job is complete, run the following commands from a Command Prompt window:

```
clusrun /exclude:%CCP_SCHEDULER% copy /y C:\Temp\%USERNAME%\Work\*.out \\%CCP_SCHEDULER%\Temp\%USERNAME%
```

```
findstr /I debug *.out
```

If the test worked correctly, this should display many lines of debug from all output files.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For more information on running Distributed Ansys under Windows HPC Server, see Configuring Distributed Ansys in the *Parallel Processing Guide*.

1.11.2. Creating a Dynamic-link (DLL) Library

You can also set up UPFs on Windows systems by using a DLL library. All Fortran files (files ending with `.F`), C files (files ending with `.c`), and C++ files (files ending in `.cpp`) that you want to include in your DLL library should reside in your working directory. To compile all `*.F`, `*.c`, and `*.cpp` routines, issue the following command:

```
\Program Files\Ansys Inc\v212\ansys\custom\user\<platform>\ANSUSERSHARED.bat
```

After you issue ANSUSERSHARED`.bat`, the output will display the options for building the DLL library. The first portion of the output is shown below:

```
This is the ANSYS 2021 R2 ANSUSERSHARED script.  It is used
to build a DLL of User Programmable Features for the program.

           NOTE:  The user subroutine source file(s) are expected to
                  reside in this directory.

 Enter one of the following choices to create your
 User Programmable Feature DLL:
```

The user routines that are supported by this method will appear in a list. (These are the same user routines as listed above for the **/UPF** command method (p. 140).) Enter the user routine name you wish to include. As an example, if you enter USERMAT the following output will display and prompt you to select another option:

```
You chose USERMAT
Microsoft (R) Incremental Linker Version 14.10.25027.0
Copyright (C) Microsoft Corporation.  All rights reserved.

-out:UserMatLib.dll
-def:UserMatLibex.def
-dll
-machine:AMD64
-map
-manifest
-manifestfile:UserMatLib.dll.intermediate.manifest
-defaultlib:ANSYS.lib
-defaultlib:bufferoverflowU.lib
*.obj
   Creating library UserMatLib.lib and object UserMatLib.exp

      ************************************************************************

      UserMatLib.dll has been successfully built.

      Set the environment variable ANS_USER_PATH to the directory where the
      UserMatLib.dll resides and run ansys212 to use your newly generated
      user shared library.

      ************************************************************************
```

After you have selected all desired user routines, press **Enter** to quit.

If you are running in a distributed memory parallel environment (Distributed Ansys), you need to include the name of the head compute node when specifying the working directory:

```
set ANS_USER_PATH=\\headnode\workingdirectory
```

When you run a user-linked version of Mechanical APDL, the output echos the value of ANS_USER_PATH and will include the following:

```
NOTE:  This Mechanical APDL version was linked by Licensee
```

To return to the original version of Mechanical APDL, return the **ANS_USER_PATH** environment variable to its original value.

Multiple UPF DLLs can be created via the ANSUSERSHARED.bat script but must exist in the same directory as designated by the **ANS_USER_PATH** environment variable.

1.11.3. Using the ANS_ADMIN Utility

The **ANS_ADMIN** procedure for compiling and linking UPFs on Windows Systems creates a custom executable. This executable can be used to run in a shared memory parallel (SMP) environment or a distributed memory parallel environment (Distributed Ansys).

As mentioned previously, the source files for the user routines reside in subdirectory `Program Files\ANSYS Inc\V212\ansys\customize\user\`.

If you modify any of the user routines, move them to the folder(s) they are linking in. By default this folder is:

`Program Files\Ansys Inc\V212\ansys\custom\user\<platform>`

Where <*platform*> is a directory that uniquely identifies the hardware platform version: "Winx64" for 64-bit Windows.

Note:

The user routines listed in Using the **/UPF** Command (p. 140) cannot be linked with the **ANS_ADMIN** utility or the ANSCUST.bat script. These user routines have reserved DLL names and must be built using either the **/UPF** command or the ANSUSERSHARED.bat script.

You can select the **Relink ANSYS** option from the **ANS_ADMIN212** utility to link these changes into Mechanical APDL. This procedure will ask which versions you intend to relink and then will compile all Fortran files (files ending with .F), C files (files ending with .c), and C++ files (files ending in .cpp) in the `Program Files\ANSYS Inc\V212\ansys\custom\user\<platform>` directory. The procedure then loads all object files (files ending with .obj), along with the default Mechanical

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

APDL objects and libraries and creates custom executables. The executable file(s) created will be named ansys.exe and will reside in the folders described above.

Caution:

When creating custom executables, they must be named ansys.exe. This requirement is due to shared library usage.

Note:

On any Windows system, if you are attempting to create a relinked version of Mechanical APDL by using ANSCUST instead of using the **ANS_ADMIN212** utility (as recommended above), error messages may occur. These messages may state that object files have been created, but the Mechanical APDL executable has not been created; or the errors may state that some libraries necessary to complete the link cannot be found. These errors usually indicate that required compiler environment variables are not set. To avoid these errors, use the following workaround when relinking Mechanical APDL with ANSCUST:

- Pick **Start > All apps > Visual Studio 2017 > Developer Command Prompt for VS2017**, which should open a new command prompt window.
- In this command prompt window, issue the drive letter and all the necessary cd commands to move to the directory where the customized files exist (example: C:\Program Files\ANSYS Inc\V212\ansys\custom\user\).
- Type ANSCUST in this command window. The process of creating the new user-customized Mechanical APDL version begins.

After relinking the Mechanical APDL executable, the program can be executed by either of the following two methods:

1. To execute the relinked version of the Mechanical APDL program:
   - Click **Start>Programs>ANSYS 2021 R2>Mechanical APDL Product Launcher**
   - In the launcher, select the **Customization/Preferences** tab, then browse to the path which contains the relinked ansys.exe. Select other desired options then pick **Run** to execute the customized ansys.exe.
2. To execute the relinked ansys.exe from a Command Prompt window, use one of the following commands.
   - Interactive:

     ```
     ansys212 -p <product variable> -g -custom <path>
     ```

   - Batch:

     ```
     ansys212 -b -p <product variable> -j jobname -i <input file> -o <output file> -custom <path>
     ```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where "path" indicates the full path to the relinked `ansys.exe`.

Note:

The `-custom` option must be the last option at the end of the command line.

Note:

Output from a user-linked version contains the following statement:

```
This Mechanical APDL version was linked by Licensee
```

1.12. Activating UPFs

The Mechanical APDL program activates many UPFs via a specific user action. This can be through a command option or a user selection. Below is a list of specific actions required for several types of UPF.

- To activate user elements created using the method described in Creating a New Element via the User-Defined Element API (p. 153), you need **USRELEM** and **USRDOF** commands, as well as **ET** and **TYPE** commands.
- To activate a user element created using the method described in Creating a New Element by Directly Accessing the Program Database (p. 171), you must select it as one of the element types in a model using the **ET** command, then set the element attribute pointer using the **TYPE** command, and define elements using the solid modeling or direct generation method.
- To define a user material described in Subroutine UserMat (Creating Your Own Material Model) (p. 205), Subroutine UserCreep (Defining Creep Material Behavior) (p. 227), and Subroutine userswstrain (Defining Your Own Swelling Laws) (p. 234), you need to activate it with the corresponding **TB** commands.
- To customize contact interfacial behaviors as described in Subroutine USERFRIC (Writing Your Own Friction Laws) (p. 242) and Subroutine USERINTER (Writing Your Own Contact Interactions) (p. 244), you need to activate them with the corresponding **TB** commands.
- To program history-dependent contact properties described in Subroutine USERCNPROP (Defining Your Own Real Constants for Contact Elements) (p. 238), you need to activate the user routine with the **R**, **RMORE**, or **RMODIF** command. The real constant must be defined by the Mechanical APDL reserved table name _CNPROP and enclosed in % signs (that is, %_CNPROP%).

UPFs that are not activated by the means described above must be activated by either of the following methods:

- Issuing the **USRCAL** command
- Choosing menu path **Main Menu>Preprocessor>Loads>-Load Step Opts->Other>User Routines** or **Main Menu>Solution>-Load Step Opts->Other>User Routines**.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To activate or deactivate the routines, issue the command **USRCAL**,*Rnam1*, ...*Rnam9*, where *Rnam1* and *Rnam9* are the names of specific routines. You can specify up to nine routines with one **USRCAL** command, or you can issue multiple **USRCAL** commands.

Issue the command **USRCAL**,NONE to deactivate all valid user subroutines. To list the status of the routines, issue the command **USRCAL**,STAT.

For a list of the user routines that the **USRCAL** command (or its equivalent menu paths) affects, see the **USRCAL** command description in the *Command Reference*.

If you do not activate the UPFs in this manner, standard Mechanical APDL logic is used by default. For example, when you apply a convection load, standard Mechanical APDL logic is the default even if you have a user convection routine linked in. The user convection routine must be activated by the **USRCAL** command or its menu equivalent.

1.13. Running Your Custom Executable

You can run a custom executable from the **Customization/Preferences** tab of the launcher:

Enter the full pathname to the custom executable in the **ANSYS Custom Executable** field. Do not include the `-custom` argument.

When run from the command prompt, if no path is specified after the `-custom` argument, the `ansys212` script searches the current working directory for the custom Mechanical APDL executable (`ansyscust.e` by default on Linux or `ansys.exe` on Windows). If the custom Mechanical APDL executable resides in a separate directory (or has a name other than `ansyscust.e` on Linux), you can specify a different path and filename after the `-custom` argument.

Caution:

If you are running on a Windows system and you create a custom Mechanical APDL executable, it must be named `ansys.exe`. This requirement is due to shared library usage.

On Linux, you can also run your custom executable via command line.

```
ansys212 -custom /pathname/ansyscust.e
```

1.14. Verifying Your Routines

After compiling and linking your new user routine, test and verify it using whatever procedures you think are adequate. Remember, verifying that your customized version of Mechanical APDL works properly is *your* responsibility.

Make certain that your custom version of the program performs correctly for the combinations of elements, analysis types, materials, boundary conditions, and so on that you plan to use. Confirm that the logic you introduced is correct and does not produce any unwanted side effects.

In testing your custom user routines, you also should verify that the changes you have made do not affect standard, non-customized Mechanical APDL features. To do so, you can compare the results of

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a set of problems from the *Mechanical APDL Verification Manual* run on the standard version and on the customized version. Input for these problems is also available on your product distribution medium.

Always remember: your last step, a series of steps, or even your concept may be wrong. Proceed in clear steps, and verify your work as often as possible. Keep intermediate versions of your modified source code on backup media.

Note:

If you contact your site's Ansys system-support person or any Ansys, Inc. representative about the performance of a custom version of Mechanical APDL, always tell that person that you are using a user programmable feature. If you feel that an error exists in an unrelated feature of Mechanical APDL, demonstrate the suspected error in a non-customized, production version of the program before you report the error to an Ansys, Inc. representative.

1.15. Debugging Commands

To debug errors in your user routines, you can use commands not documented in the *Command Reference*. Use these commands only for extremely small models with few solution iterations (otherwise, they will generate an excessive amount of output).

Two useful commands are **OUTEQ** and **/NERR**. The command **OUTEQ**,ON can be used to output results from all equilibrium iterations. The command **/NERR**,,,-1 causes errors to be reported as before, but the run continues anyway, normally terminating with either a) system abort or b) incorrect answers. The **/NERR**,,,-1 command is intended for program debugging and may generate erroneous results. You should remove this statement before generating solutions for production use.

1.16. Other Useful Commands

Two other Mechanical APDL commands, **NSVR** and **/UCMD**, can help you implement UPFs. (Neither command has an equivalent GUI path.) Use the **NSVR** command to define the number of extra variables that need to be saved for user programmable element options, such as user plasticity.

Issue the **/UCMD** command to make a user routine into a custom command. For more information, see Defining Your Own Commands (p. 267).

1.17. Generating Output

You can generate output controlled by the **/OUTPUT** command by using the Fortran write statement. The output unit for this statement is usually called IOTT. IOTT may be defined with the function `wrinqr`. See the discussion on the function `wrinqr` in Subroutines for Your Convenience (p. 351) for more details.

1.18. Reading Large Data Files More Rapidly

When files containing Mechanical APDL-related data are large, loading them into the program or writing them out to an external file can be a slow process. For example, consider a problem file which contains nearly 462,000 lines, 150,000 of which contain nodal data and 97,383 of them containing data for ele-

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ments. Because many of the lines in this file are in command format, Mechanical APDL requires much time to read it.

You can shorten the time the program requires to read such files by including two commands in your programs, UPFs, or macros: EBLOCK (p. 113) and NBLOCK (p. 117). The NBLOCK (p. 117) command converts nodal data into fixed format data blocks (which Mechanical APDL can read more quickly than commands). The EBLOCK (p. 113) command places element data into a fixed format block, one line per element. These commands also compress displacement constraint data to one line per constraint node. See The CDWRITE (CDB) File Format (p. 99) in the *Guide to Interfacing with ANSYS* for more information on the use of these commands.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 2: UPF Subroutines and Functions

This guide describes the various subroutines, functions, and commands that allow you to customize the program for your specific purpose. The first portion of each subroutine or function (consisting of comment lines) is shown in most cases.

User subroutines that can be modified have the term *USERDISTRIB* in the first line of the subroutine. For example, the first line of the `usercnprop` subroutine looks like this:

```
*deck,usercnprop                    USERDISTRIB
```

User subroutines that do not have *USERDISTRIB* in the first line cannot be modified and must be used as they are.

The following UPF topics are available:

2.1. Creating a New Element

Two tools are available for creating a user-defined element:

- The user-defined element API (p. 153)
- Direct access to the program database and files (p. 171)

Ansys, Inc. recommends the user-defined element API in most cases. The direct-access method is generally for special-purpose use only, or if you are already using preexisting elements created with this method.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This table highlights the differences between the two methods:

Interface	User-defined element API	Accessing program database and files directly
Description	Offers a simpler interface while preserving much of the underlying user-element capability. An understanding of the database subroutines and the file structure is rarely necessary to use the interface.	No special interface. With few exceptions, if a capability exists for an element, it will exist here. The logic necessary for using this interface effectively is more complex.
Relative level of difficulty	Medium	High
Expected compatibility between versions	High	Medium
Element names	USER300	USER100 to USER105
Demonstration logic included on your product distribution media	4-node quad and 20-node brick elements	`uel100` (a structural mass element) and `uel101` (a simple link element)
Typical linear material access subroutine	`getMatProp`	`propev`
New nonlinear material properties	Program in `UserMatTh`.	No special programming has been set up.
Existing nonlinear material properties	All standard structural materials are accessible via `ElemGetMat`.	Limited capability. Accessible via `plastx`, `creepx`, and `swellx`.
Non-structural material properties	No special programming has been implemented.	No special programming has been implemented.
Number of different element types allowed	Effectively, no limit.	Effectively, no limit.
Element characteristic capability	Input the basic 10 element characteristics (via the **USRELEM** and **USRDOF** commands). All other characteristics default automatically.	Input all 140 element characteristics (using `uec100`). Approximately 30 are normally used, and the rest default unless required for special cases.
Applicable subroutines to be programmed	`UserElem`	`uec100`, `uel100`, and rarely `uex100` and `uep100`. Subroutines `uec101` to `uec105` are analogous.
Access to database information	Generally through the interface.	Through subroutines (such as `tmpget`, `prsget`, `svgidx`, `svrget`, `svpidx`, `svrput`).
Access to file information	Through the interface.	Through pointers and subroutines (such as `eldwrtL`, `eldwrnL`).
Special features	Element convergence criteria Cutback control via element	None.
Capabilities *not* included	Control information unavailable for:	Material **TB** labels PLASTIC, NLISO, AHYPER, HYPER, PRONY, SHIFT,

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Interface	User-defined element API	Accessing program database and files directly
	Birth and death Superelement stress pass Initial stress Section input Input of fluences Swelling	ELASTIC, ANEL, SDAMP, SMA, CAST, EDP, and GURSON.

2.1.1. Input and Output Abbreviations

The descriptions of the subroutines or functions within this chapter describe both the input arguments and output arguments. Argument information includes the argument's type, size and intent.

- Argument *type* is one of the following:

  int - integer
  dp - double-precision
  log - logical
  chr - character
  dcp - double-precision complex

- Argument *size* is one of the following:

  sc - scalar variable
  ar(*n*) - array variable of length *n*
  func - functional return value

- Argument *intent* is one of the following:

  in - input argument
  out - output argument
  inout - both an input and an output argument

2.1.2. Creating a New Element via the User-Defined Element API

Following is the general process for creating your own element via the user-defined element API.

Steps 2 and 3 specify data for the user-defined element API. All other steps represent standard features.

Step	Description	Comments
1.	Specify the element type.	Issue the **ET** and **TYPE** commands. The name of the element must be USER300.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.	Define your new element according to the specified element type.	Issue the **USRELEM** command. Specify the element characteristics (such as the number of nodes, number of dimensions, number of real constants etc.).
3.	Specify nodal degrees of freedom.	Issue the **USRDOF** command. You can specify a maximum of 10 degrees of freedom per **USRDOF** command; to define additional degrees of freedom, issue the command again. Each node will have the same degrees of freedom. Although you can specify any valid degrees of freedom, the total number of degrees of freedom for your element cannot exceed 480, and the number of degrees of freedom for each node cannot exceed 32.
4.	Define real constants.	If needed.
5.	Create finite element models.	Use either of these methods: • *Direct generation* -- Create elements directly from nodes, using commands such as **E**, **EGEN**, **EN**, **ENGEN**, or **EMORE**. (You can also use the **CDREAD** command if the `.cdb` file is available.) This method is the only way to create an element with a topology different from that of any standard element. • *Meshing commands* -- This method is available only if your element has the same topology as that of a standard element *and* you have specified any standard element shape (**USRELEM** *KeyShape* value) except ANYSHAPE.
6.	Apply boundary conditions and loads.	As needed.
7.	Specify solution options.	If your element has multi-field degrees of freedom (displacements and temperatures).
8.	Perform postprocessing.	Postprocessing occurs normally as with any other element. Only total strain (or equivalent quantities such as thermal gradient) and stress (or equivalent quantities such as thermal flux) are saved as regular result quantities. Other variables are saved as nonsummable miscellaneous variables in the results file.

Recommendations and Restrictions

The following recommendations and restrictions apply to user-defined element USER300:

- Verify that your input data for the **USRELEM** and **USRDOF** commands are consistent with the values used in the `UserElem.F` code. For example, if the number of dimensions (*NDIM*) specified via the **USRELEM** command is 2, do not change the number of dimensions specified in the `UserElem.F` subroutine from 2. A runtime error or incorrect results can occur if the values do not match.
- The program may activate default solution settings automatically according to the USER300 element's degrees of freedom, but the default solution control settings may not be optimal for your element.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The USER300 element does not support section (SEC*xxx*) commands. For composite beams and layered shells, you must input element data via real constants and code the `UserElem.F` subroutine accordingly.

2.1.2.1. Subroutine UserElem (Writing Your Own Elements)

The `UserElem` subroutine provides an interface to program code above the element level. `UserElem` supports shared memory and distributed parallel processing; however, you are responsible for ensuring that your code can use parallel processing.

The subroutine passes all data needed to create a user-defined element and returns all data and results from the element to update the program database and files. With this API, you can create virtually any element type *without* having to access program database and files directly (p. 171). Two examples are included in this subroutine: a 4-node quadrilateral 2-D element, and 20-node brick structural element, both for geometric linear analysis. Key options (KEYOPT settings) switch the elements.

The following table shows the input and output arguments, and their definition and usage. Some argument names (such as those pertaining to element matrices and load vectors) are common to structural analyses; however, you can specify argument values appropriate to analyses in other engineering disciplines.

Argument	Input (I) *or* Output (O)	Definition	Purpose	How Defined
`elId`	I	Element number	Output information	At FE model creation
`matId`	I	Material number	Output information Call material subroutines	At FE model creation
`keyMtx`	I	Formulation request	Specifying which matrices and load vectors to form	Program code
`lumpm`	I	Mass matrix format: 0 = Consistent mass matrix 1 = Lumped mass matrix	Specifying how to form the mass matrix	**LUMPM** command
`nDim`	I	Number of dimensions	Element coding	**USRELEM** command
`nNodes`	I	Number of element nodes	Element coding	**USRELEM** command
`Nodes`	I	Element node list Connectivity	Output	At FE model creation

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

nIntPnts	I	Maximum number of element integration points	Element coding	**USRELEM** command
nUsrDof	I	Number of element degrees of freedom	Element coding -- The degrees of freedom are ordered in the way in which they are listed via the **USRDOF** command for each node and repeated for all nodes All element matrices -- DOF values and load vectors must be arranged in the same way	**USRELEM** and **USRDOF** commands
kEStress	I	Element stress state	Element coding Calling material subroutines if requested	**USRELEM** command
keyAnsMat	I	Element formulation key: 0 -- Write your own material formulation 1 -- Use standard material subroutines and call `ElemGetMat` subroutine	Specifying how to create material data	**USRELEM** command

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

keySym	I	Flag for symmetricity of element matrices	Element coding Program assembly logic	**USRELEM** command
nKeyOpt	I	Maximum number of element key options allowed (up to 2)	Element coding	Program code
KeyOpt	I	Element key options KEYOPT(1) = 0~99	Branching the user-element codes to different formulations. (This could be equivalent to 100 x 100 different types of elements.)	**ET** command
temper	I	Nodal temperatures at current time	Temperature dependence and thermal loads	**BF** and **BFE** commands (if **keyShape** is specified in the `UserElem` subroutine)
temperB	I	Nodal temperatures at the end of the last substep	Temperature dependence and thermal loads	Program code
tRef	I	Reference temperature	Temperature dependence and thermal loads	**TREF** command
kTherm	I	Key indicating whether a thermal load exists: 1 = Calculate the thermal load 0 = No thermal load calculation	Element coding	---
nPress	I	Number of pressure values	Element coding The size of the press vector	**USRELEM** command At FE model creation

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Press	I	Pressures at nodes of element facets (available only when **keyShape** is specified via the **USRELEM** command) The pressure vector is ordered in the element with the same topology as that in the standard element library. Refer to that element for details.	Pressure load and pressure load stiffness	**SF** and **SFE** commands
kPress	I	Key indicating whether a pressure load exists: 1 = Calculate the pressure load 0 = No pressure load calculation	Element coding	Program code
nReal	I	Number of real constants	Element coding	**USRELEM** command
RealConst	I	The list of real constants	Element coding Can pass material properties, section/layer information, element material control, and any element data	**R** command
nSaveVars	I	The number of variables saved in the `.esav` file for the element	Element coding The size of **saveVars**	**USRELEM** command
saveVars	I/O	The data saved in the `.esav` file The program saves the data after	Element coding	`UserElem` subroutine

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

		exiting the `UserElem` subroutine and retrieves it immediately before entering `UserElem` again. It should include kinematic related variables only when the material subroutine is called; otherwise, it should include both kinematic and material data. History dependent variables can only be saved/updated when the substep is converged (**`keyHisUpd`** = 1).		
`xRef`	I	Initial coordinates of the element nodes Values in global Cartesian coordinates	Element coding	At FE model creation
`xCur`	I	Current (deformed) coordinates of element nodes Values in global Cartesian coordinate system, equal to **`xRef`** when **`nlgeom`** = off	Element coding	Program code
`TotValDofs`	I	Total values of degrees of freedom (displacements for structural analysis) Values in global Cartesian coordinate system	Element coding	Program code
`IncValDofs`	I	Increment values of degrees of	Element coding	Program code

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

		freedom occurring at the current substeps Values in global Cartesian coordinate system		
ItrVal-Dofs	I	Iteration values of degrees of freedom occurring at the last iteration Values in global Cartesian coordinate system	Element coding	Program code
VelVal-Dofs	I	First time derivatives of degrees of freedom	Velocities	Program code
AccVal-Dofs	I	Second time derivatives of degrees of freedom	Accelerations	Program code
kfstps	I	Key indicating first time entering the element subroutine: 1 = First time 0 = Other than first time	Initializing data	Program code
nlgeom	I	Flag indicating whether large displacement/deformation is in effect	Element coding	**NLGEOM** command
nrkey	I	Newton-Raphson algorithm key: 1 -- Any nonlinear analysis 0 -- Pure linear analysis	Output Element coding	---
outkey	I	Key indicating output result type: 1 -- This is an output call, the substep is converged, and you can print/save element results 0 -- All other cases	Element coding	Program code

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

elPrint	I	Key indicating whether any element output should appear in the print file: 0 = No 1 = Yes	Element coding	Program code **OUTPR** command
iott	I	Output file number	The FORTRAN output file number. All information written in the specified file appears in the output file.	Program code
keyHisUpd	I	Key to update history-dependent variables: 1 = The substep converged; ready to update history-dependent variables (such as equivalent plastic strain) 0 = Solution not yet converged; cannot update history-dependent variables	Element coding	Program code
The following variables are for debug, timing, and convergence-control purposes only. You can usually ignore them.				
ldstep	I	Current load step number	Output Debug	Program code
isubst	I	Current substep number	Output	Program code
ieqitr	I	Current iteration number	Output	Program code
timval	I	Current time	Output	Program code
keyEleErr	I/O	Formulation error key: 0 = No error (preset value) 1 = Error occurred in element	Element coding	Program code

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

		formulation, possibly due to excessive deformation. (The program lessens deformation if possible by cutback.)		
keyEleCnv	I/O	Element convergence key: 1 = Converged (preset value before calling) 0 = Not converged	Provides manual control of convergence when you introduce any constraint at the element level (such as volumetric constraint for mixed u-P)	Program code
End of special-purpose variable group				
eStiff	O	Small-deformation stiffness matrix In global Cartesian coordinate system	Solution	Requested when **keyMtx(1)** = 1
eMass	O	Mass matrix In global Cartesian coordinate system	Solution	Requested when **keyMtx(2)** = 1
eDamp	O	Damping matrix In global Cartesian coordinate system	Solution	Requested when **keyMtx(3)** = 1
eSStiff	O	Stress stiffness matrix In global Cartesian coordinate system	Solution	Requested when **keyMtx(4)** = 1
fExt	O	External load vector In global Cartesian coordinate system	Solution	Requested when **keyMtx(5)** = 1
fInt	O	Internal nodal force vector In global Cartesian coordinate system	Solution	Requested when **keyMtx(6)** = 1
elVol	O	Element volume	Output	`UserElem` subroutine

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

elMass	O	Element mass	Output	UserElem subroutine
elCG	O	Element centroid coordinates In global Cartesian coordinate system	Postprocessing	UserElem subroutine
nRsltBsc	I	Number of basic result data saved in result file	Specifying the size of **RsltBsc**	Program code
RsltBsc	O	Basic result data saved in results file These variables are accessible via the **PRESOL** and **PRNSOL** commands in the standard way and can also be plotted if you specify a *KeyShape* value via the **USRELEM** command.	Postprocessing	UserElem subroutine
nRsltVar	I	The number of result data to be saved in the result file as non-summable miscellaneous variables	Specifying the size of **RsltVar**	**USRELEM** command
RsltVar	O	The result data saved in the result file as non-summable miscellaneous variables The data is accessible via the **PLESOL** command only, but only one value for an element each time	Postprocessing	UserElem subroutine
nElEng	I	Number of energies Fixed at 3	Solution	UserElem subroutine
elEnergy	O	Element energy vector: **elEnergy(1)** -- Strain energy	Output	UserElem subroutine

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

		elEnergy(2) -- Plastic energy **elEnergy(3)** -- Creep energy		

```
*deck,UserElem                      USERDISTRIB
c Copyright ANSYS.  All Rights Reserved.
      subroutine UserElem (elId, matId, keyMtx, lumpm, nDim, nNodes,
     &                     Nodes, nIntPnts, nUsrDof, kEStress,
     &                     keyAnsMat, keySym, nKeyOpt, KeyOpt,
     &                     temper, temperB, tRef, kTherm,
     &                     nPress, Press, kPress, nReal, RealConst,
     &                     nSaveVars, saveVars, xRef, xCur,
     &                     TotValDofs, IncValDofs, ItrValDofs,
     &                     VelValDofs, AccValDofs,
     &                     kfstps, nlgeom, nrkey, outkey, elPrint, iott,
     &                     keyHisUpd, ldstep, isubst, ieqitr, timval,
     &                     keyEleErr, keyEleCnv,
     &                     eStiff, eMass, eDamp, eSStiff,
     &                     fExt, fInt, elVol, elMass, elCG,
     &                     nRsltBsc, RsltBsc, nRsltVar, RsltVar,
     &                     nElEng, elEnergy)
     &
c*************************************************************************
c
c *** Primary function: General User Element Subroutine
c *** Note:
c       This routine is completed with an example, see more details later.
c
c
c     PROGRAMMER SHOULD NOT CHANGE ANY PURE INPUT ARGUMENTS (marked by ....,in)!
c
c     elId      (int,sc,in)        element number
c     matId     (int,sc,in)        material number of this element
c     keyMtx    (int,ar(10),in)    matrix and load vector form requests
c                                     0 = not requested, 1 = requested
c                                     see below for more details
c     lumpm     (int,sc,in)        mass matrix format
c                                    = 0 no lumped mass matrix
c                                    = 1 lumped mass matrix
c     nDim      (int,sc,in)        number of dimensions of the problem
c                                        (defined on USRELEM command as NDIM)
c                                    = 2 2D
c                                    = 3 3D
c     nNodes    (int,sc,in)        number of nodes of the element
c                                        (defined on USRELEM command as NNODES)
c     Nodes     (int,ar(nNodes),in)node list of this element
c     nIntPnts  (int,sc,in)        maximum number of integration points
c                                        (defined on USRELEM command as NINTPNTS)
c     nUsrDof   (int,sc,in)        number of DOFs of this element (matrix and
c                                     load vector size)
c     kEStress  (int,sc,in)        kEStress
c                                        (defined on USRELEM command as KESTRESS)
c     keyAnsMat (int,sc,in)        key to indicate if ANSYS material
c                                     routine is going to be called
c                                     (defined on USRELEM command as KEYANSMAT)
c                                     = 0, No
c                                     = 1, Yes
c     keySym    (int,sc,in)        key to indicate if element matrices
c                                     is symmetric
c                                       (defined on USRELEM command as KEYSYM)
c                                     = 0, symmetric
c                                     = 1, unsymmetric
c     nKeyOpt   (int,sc,in)        number of element key options able to be
c                                     used in this routine
c     KeyOpt    (int,ar(nKeyOpt),in) values of element key option defined
c                                     by et or keyopt command for the
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                     user elements, only the first
c                                     nKeyOpt values are passed in and can
c                                     be used to branch the routine for
c                                     different formulations
c     temper    (dp,ar(nNodes),in) nodal temperatures at current time
c     temperB   (dp,ar(nNodes),in) nodal temperatures at the beginning of this
c                                     incremental step (substep)
c     tRef      (dp,sc,in)         reference temperature
c     kTherm    (int,sc,inout)     input:  flag for thermal loading
c                                      = 1, Temperatures at nodes are different
c                                      from the reference temperature,
c                                      thermal loading might be needed.
c                                      = 0, Temperatures at nodes are the same
c                                      as the reference temperature,
c                                      thermal loading is not needed.
c                                    output:  flag for thermal strains
c     nPress    (int,sc,in)        number of pressure values for this element
c     Press     (dp,ar(nPress),in) applied elemental face load (pressure)
c     kPress    (int,sc,in)        flag for pressure loading
c                                      = 1, pressure load is applied and
c                                      equivalent nodal forces should be
c                                      calculated
c                                      = 0, no pressure loading
c     nReal     (int,sc,in)        number of real constants
c                                       (defined on USRELEM command as NREAL)
c     RealConst (dp,ar(nReal),in)  user defined real constants
c     nSaveVars (int,sc,in)        number of saved variables
c                                      (defined on USRELEM command as NSAVEVARS)
c     saveVars  (dp,ar(nSaveVars),inout) user saved variables
c     xRef      (dp,ar(nDim,nNodes),in)
c                                    nodal coordinates in initial configuration
c     xCur      (dp,ar(nDim,nNodes),in)
c                                    nodal coordinates in current configuration
c     TotValDofs (dp,ar(nUsrDof),in) total values of DOFs (displacements)
c                                     from time = 0
c     IncValDofs (dp,ar(nUsrDof),in) incremental values of DOFs (displacements)
c                                     for the current step
c     ItrValDofs (dp,ar(nUsrDof),in) iterative values of DOFs (displacements)
c                                     for the current iteration
c                                     (normally needed for debug only)
c     VelValDofs (dp,ar(nUsrDof),in) first time derivatives of DOFs
c                                              (velocities) (normally not needed)
c     AccValDofs (dp,ar(nUsrDof),in) second time derivatives of DOFs
c                                           (accelerations) (normally not needed)
c     kfstps    (int,sc,in)        key for the first iteration of first
c                                     substep of the first load step
c                                     = 1 yes
c                                     = 0 no
c     nlgeom    (int,sc,in)        large deformation key [from nlgeom command]
c                                     = 0 NLGEOM,OFF
c                                     = 1 NLGEOM, ON
c     nrkey     (int,sc,in)        key to indicate a newton-raphson
c                                     (incremental) procedure
c                                     = 0 No
c                                     = 1 Yes
c     outkey    (int,sc,in)        key to indicate if any element output is
c                                     to be placed on the print file or the
c                                     result file
c                                     = 0 No
c                                     = 1 Yes
c     elPrint   (int,sc,in)        key to indicate if any element output is
c                                     to be placed on the print file
c                                     = 0 No
c                                     = 1 Yes
c     iott      (int,sc,in)        print output file unit number
c     keyHisUpd (int,sc,in)        key to indicate if history-dependent
c                                    variables need to be updated, like
c                                    equivalent plastic strain, back stress
c                                    etc. since the iteration is already
c                                    converged
c                                     = 0 not converged, don't need to update
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                       history dependent variables
c                                    = 1 yes, converged, need to update
c                                       history dependent variables
c
c     --- The following 7 variable group can usually be ignored.
c     --- The variables are used for debug, timing, and convergence control.
c     ldstep    (int,sc,in)        current load step number
c     isubst    (int,sc,in)        current substep number
c     ieqitr    (int,sc,in)        current equilibium iteration  number
c     timval    (int,sc,in)        current time value
c     keyEleErr (int,sc,inout)     key to indicate if there is any element
c                                     formulation error, like negative Jacobian.
c                                     The error could be caused by too
c                                     large incremental step, illegal model.
c                                     = 0 no error (preset value before calling)
c                                     = 1 some error happens. ANSYS will
c                                     decide to stop the analysis or cutback
c                                     the substep (bi-section) based on other
c                                     user input and information at higher
c                                     level.
c     keyEleCnv (int,sc,inout)     key to flag if this element satisfies
c                                     the user defined element convergence
c                                     criterion.
c                                     = 1, yes, the criterion is satisfied
c                                       or don't have any criterion at all
c                                       it is preset value before calling
c                                     = 0, no, the element doesn't satisfy
c                                       element convergence criterion. If
c                                       this is the case, the iteration will
c                                       not converge even when both force
c                                       and displacement converge
c       ---- end of 7 variable group -----
c
c                                                                    requested if
c     eStiff(dp,ar(nUsrDof,nUsrDof),inout) stiffness matrix         keyMtx(1)=1
c     eMass (dp,ar(nUsrDof,nUsrDof),inout) mass matrix              keyMtx(2)=1
c     eDamp (dp,ar(nUsrDof,nUsrDof),inout) damping matrix           keyMtx(3)=1
c     eSStiff(dp,ar(nUsrDof,nUsrDof),inout)stress stiffness matrix  keyMtx(4)=1
c     fExt      (dp,ar(nUsrDof),out)       applied f vector         keyMtx(5)=1
c     fInt      (dp,ar(nUsrDof),out)       internal force vector    keyMtx(6)=1

c     elVol     (dp,sc,out)        element volume
c     elMass    (dp,sc,out)        element mass
c     elCG      (dp,ar(3),out)     element centroid coordinates in current
c                                     configuration
c     nRsltBsc  (dp,sc,in)         number of basic elemental results saved in
c                                   result files
c     RsltBsc   (dp,ar(nRsltBsc),out) basic elemental results
c                                       (see EXPLANATION below)
c     nRsltVar  (int,sc,in)        number of elemental results saved in
c                                     result file as non-summable miscellaneous
c                                     variables
c                                       (defined on USRELEM command as NRSLTVAR)
c     RsltVar   (dp,ar(nRsltVar),out) variables to saved in result files as
c                                      non-summable miscellaneous variables
c                                      requested when outkey = 1
c
c     nElEng    (int,sc,in)        number of energies (fixed at 3)
c
c     elEnergy  (dp,ar(nElEng),out) elemental energy
c                                     elEnergy(1), element strain energy
c                                     elEnergy(2), element plastic strain energy
c                                     elEnergy(3), element creep strain energy
c
c     EXPLANATION OF RsltBsc
c
c       Basic element results are saved and total number of result
c     quantities is nRsltBsc, where:
c            nRsltBsc = (7+7)* number of corner nodes at one element.
c       To process the quantites by post processing properly, the results
c     must be in the following order:
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c       1.) Stresses: Sx Sy Sz Sxy Syz Sxz Seqv at all corner points,
c     followed by:
c       2.) Strains : Ex Ey Ez Exy Eyz Exz Eeqv at all corner points
c     where Seqv and Eeqv = equivalent stress and strain respectively
c
c
**********************************************************************
c
```

2.1.2.2. Subroutine ElemGetMat (Calling the Standard Structural Material Library)

The `ElemGetMat` subroutine is the API to the standard materials. When you issue the **USRELEM** command after setting the command's *KEYANSMAT* argument, the subroutine accesses the standard structural material library. It allows you to focus on the kinematic portion of element formulation while the program handles the material part of the formulation.

When calling the subroutine, input the associated material data via the standard method. There is no need to access this subroutine, only to call it.

The following table shows the input and output arguments, and their definition and usage.

Argument	Input (I) *or* Output (O)	Definition	Purpose	How Defined
`elId`	I	Element number	Output	At FE model creation
`matId`	I	Material number	Output information Getting material data	At FE model creation
`nDim`	I	Number of dimensions of element geometry 2 = 2-D element geometry 3 = 3-D element geometry Specifying the size of the nodal coordinates	Material calculation	At FE model creation
`nTens`	I	Number of stress/strain tensor components: 4 = 2-D and ordered as x, y, z, xy 6 = 3-D and ordered as x, y, z, xy, yz, xz	Specifying the data size	`UserElem` subroutine

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

nDirect	I	Number of direct component of stress/strain tensors **nDirect**< or = **nTens**	Specifying the data size	UserElem subroutine
intPnt	I	Current integration point number	Output Data handling	UserElem subroutine
xCurIP	I	Coordinates of current integration point Values in global Cartesian coordinate system	Material calculation	UserElem subroutine
TemperIP	I	Integration point temperatures at the current time	Evaluating temperature-dependent material data	UserElem subroutine
TemperIPB	I	Integration point temperatures at the end of the last incremental step	Evaluating temperature-dependent material data	UserElem subroutine
IncStrain	I	Strain components [1] Incremental strain of the current substep when **nlgeom** = on Total strain when **nlgeom** = off	Material calculation	UserElem subroutine
defG0	I	Deformation gradient tensor at the end of previous substep [1]	Material updating	UserElem subroutine
defG	I/O	Total deformation gradient tensor at the current time [1]	The component in thickness direction is updated by material subroutines for plane stress and shell elements	UserElem subroutine
kTherm	I/O	Thermal loading key: 0 = No thermal loading 1 = Has thermal loading	Thermal load calculation	UserElem subroutine
cMat	O	Material Jacobian [1]	Forming stiffness	Material subroutine

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MatProp	O	Material data for element formulation	Forming mass matrix Handling transverse shear Output	Material subroutine
Stress	O	Cauchy stress [1]	Forming geometric stiffness Calculating internal forces	Material subroutine
Strain	O	Total strain components [1]	Output	Material subroutine
StressTh	O	Total thermal stress components [1]	Output Calculating thermal loading	Material subroutine
StrainTh	O	Total thermal strain components [1]	Output	Material subroutine
StrainPl	O	Total plastic strain components [1]	Output	---
StrainCr	O	Total creep strain components [1]	Output	---
StressBk	O	Backstress components [1]	Output	---
StrainSw	O	Swelling strain	*Not yet supported*	---
EnergyD	O	Energy density: 1 = Elastic energy density 2 = Plastic energy density 3 = Creep energy density	---	---
MatRotGlb	O	Rotation matrix	Rotation matrix from global Cartesian to rotated element coordinate system	Used only for solid elements when **nlgeom** = on

1. All tensor component values in the subroutine are in the global Cartesian coordinate system for solid elements, and in the co-rotated element Cartesian coordinate system for link, beam and shell elements.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
*deck,ElemGetMat

      subroutine ElemGetMat (elId, matId,  nDim, nTens, nDirect,
     &                           intPnt, xCurIP, TemperIP,
     &                           TemperIPB, kTherm, IncStrain,
     &                           defG0, defG, CMat, MatProp,
     &                           Stress, Strain, StressTh, StrainTh,
     &                           StrainPl, StrainCr, StressBk, StrainSw,
     &                           EnergyD, MatRotGlb)

c**********************************************************************
c
c *** Primary function: call ANSYS material routines to obtain material
c                       data for USER300 elements

c *** Notice - This file contains ANSYS Confidential information ***
c
c     input arguments
c     ===============
c     elId        (int,sc)           element number
c     matId       (int,sc)           material number of this element
c     nDim        (int,sc)           number of dimensions of the problem
c                                    = 2 2D
c                                    = 3 3D
c     nTens       (int,sc)           number of stress/strain components
c     nDirect     (int,sc)           number of stress/strain direct
c                                      components
c     intPnt      (int,sc)           current integration point number
c     xCurIP      (dp,ar(3))         coordinates of integration point
c     TemperIP    (dp,sc)            integration point  temperatures at
c                                      current time
c     TemperIPB   (dp,sc)            integration point  temperatures at
c                                      the end of last incremetal step
c     IncStrain   (dp,ar(nTens))     strain for the current substep step when
c                                       nlgeom = on
c                                    total strain when nlgeom = off
c     defG0       (dp,ar(3x3))       deformation gradient tensor at the end
c                                       of last incremental step
c
c     input output arguments         input desc     / output desc
c     ======================         ==========       ===========
c     defG        (dp, ar(3x3))      deformation gradient tensor at current
c                                      time, updated for thichness change in
c                                      plane stress when nlgeom=on
c     kTherm      (int,sc)           flag for thermal loading
c                                      input as:
c                                      = 0 if temp = tref
c                                      = 1 if temp .ne. tref
c                                      gets reset to 0
c                                                  if ALPX, ALPY, and ALPZ = 0
c
c     output arguments
c     ================
c     cMat        (nTens*nTens)      material Jacobian matrix
c     MatProp     (dp,ar(5))         commonly used materail properties
c                                    MatProp(1),Gxz: shear modulus
c                                    MatProp(2),Gyz: shear modulus
c                                    MatProp(3),Gxy: shear modulus
c                                    MatProp(4), density
c                                    MatProp(5), nuxy
c     Stress      (dp,ar(nTens))     total stress
c     Strain      (dp,ar(nTens))     total strain
c     StressTh    (dp,ar(nTens))     thermal stress
c     StrainTh    (dp,ar(nTens))     thermal strain
c     StrainPl    (dp,ar(nTens))     plastic strain
c     StrainCr    (dp,ar(nTens))     creep strain
c     StressBk    (dp,ar(nTens))     back stress for kinematic hardening
c     StrainSw    (dp,sc)            isotropic swelling strain
c                                        (swelling capability not available yet)
c     EnergyD      (dp,ar(3))        energy density
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                    EnergyD(1) elastic energy density
c                                    EnergyD(2) plastic energy density
c                                    EnergyD(3) creep energy density
c     MatRotGlb   (dp,ar(3,3))       The rotation matrix from global
c                                     to  material coordinate system
*************************************************************************
c
```

2.1.3. Creating a New Element by Directly Accessing the Program Database

The next few pages describe the user subroutines and supporting subroutines you use to create new elements. Using these subroutines, you can create new element types, add them to the element library, and use them as "regular" elements. You can create up to six independent element types (names USER100 - USER105). For demonstration purposes, the subroutines `uel100` (for a structural mass element) and `uel101` (for a simple link element) are included on the product distribution media.

2.1.3.1. User Subroutines

Subroutines `uec100` through `uec105` describe the element characteristics. Subroutine `elccmt` (on the distribution medium) describes the input for these subroutines in detail. You can use subroutines `uex100` through `uex105` to override default logic. Subroutines `uec100` through `uec105` define parameters such as:

- 2-D or 3-D geometry
- Degree of freedom set
- Symmetric or unsymmetric matrix
- Number of nodes
- Number of body loads (for example, temperatures)
- Number of surface loads (for example, pressures)
- Number of real constants
- Number of variables to be saved
- Number of rows in element matrices
- Linear or nonlinear element.

Subroutines `uel100` through `uel105` calculate the element matrices (stiffness, specific heat, and so on), the element load vector (force, heat flow, and so on), and any element output quantities. The element printout also is generated, and the variables to be saved are calculated and stored in the results file.

Other user subroutines available for manipulating element information include the following:

- Subroutines `uep100` through `uep105` provide printed output of line elements. The general postprocessor, POST1, calls the subroutines, or you can call them using `uel100` through `uel105`.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Subroutine usertr allows access to the nodal transformations.
- Subroutine userac describes some of the data handling.

2.1.3.2. Subroutine uec100 (Defining Characteristics of the usr100 Subroutine)

```
*deck,uec100                      USERDISTRIB
      subroutine uec100 (elcdn,ielc,kerr)
c     ***** this subroutine defines the characteristics of user100.
c
c         *** Copyright ANSYS.  All Rights Reserved.
c         *** ansys, inc.
c *** Notice - This file contains ANSYS Confidential information ***
c
c     typ=int,dp,log,chr   siz=sc,ar(n)   intent=in,out,inout
c
c  input arguments:
c     variable (typ,siz,intent)     description
c     ielc (int,ar(IELCSZ),inout) - element characteristics
c                                        see include deck elccmt for details
c     kerr   (int,sc,inout)       - error flag up to this point.
c                                    (do not initialize to zero)
c
c  output arguments:
c     variable (typ,siz,intent)     description
c     elcdn   (chr,sc,out)        - name of element
c     ielc (int,ar(IELCSZ),inout) - element characteristics
c                                          see include deck elccmt for details
c     kerr   (int,sc,inout)       - error flag (set to 1 if error)
c       note to programmers:  the validity of keyopt values may be checked here
c
```

2.1.3.2.1. Subroutines uec101 through uec105

The input and output arguments for subroutines uec101, uec102, uec103, uec104, and uec105 is identical to the uec100 subroutine listed above.

2.1.3.3. Subroutine uex100 (Overriding Element Characteristic Defaults)

```
*deck,uex100                      USERDISTRIB
      subroutine uex100 (ielc,kerr)
c     *** subroutine to override element characteristic defaults ***
c     *** hence, this routine is needed only in rare cases.
c
c         *** Copyright ANSYS.  All Rights Reserved.
c         *** ansys, inc.
c *** Notice - This file contains ANSYS Confidential information ***
c
c     *** input and output are the same as for uec100, except that this
c     *** logic is called after the defaulting logic is finished.
c     *** this defaulting is done in ansys subroutine echdft(not a upf).
c     *** as indicated above, this routine is rarely needed, but if it is
c     *** desired to see the effect of echdft, you may print out the ielc array
c     *** leaving uec100 and print it out again entering this routine.
c
c     typ=int,dp,log,chr   siz=sc,ar(n)   intent=in,out,inout
c
c  input arguments:
c     variable (typ,siz,intent)     description
c     ielc (int,ar(IELCSZ),inout) - element characteristics
c                                        see include deck elccmt for details
c     kerr   (int,sc,inout)       - error flag up to this point.
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                (do not initialize to zero)
c
c  output arguments:
c     variable (typ,siz,intent)     description
c     ielc (int,ar(IELCSZ),inout) - element characteristics
c                                         see include deck elccmt for details
c     kerr   (int,sc,inout)       - error flag (set to 1 if error)
c
```

2.1.3.3.1. Subroutines uex101 through uex105

The source code for subroutines `uex101`, `uex102`, `uex103`, `uex104`, and `uex105` is identical to the code for subroutine `uex100` listed above.

2.1.3.4. Subroutine uel100 (Calculating Element Matrices, Load Vectors, and Results)

```
*deck,uel100                      USERDISTRIB
      subroutine uel100 (elem,ielc,elmdat,eomaskL,nodes,locsvrL,kelreq,
     x kelfil,nr,xyz,u,kelout,zs,zass,damp,gstif,zsc,zscnr,elvol,elmass,
     x center,elener,edindxL,lcerstL)
c --- general lumped mass is demonstrated --------------------------------
c *** primary function:
c        1. compute element matrices, load vectors, and results
c *** secondary functions:
c        2. maintain element solution data

c *** user programmable functions may not be used in parallel processing ***

c *** Notice - This file contains ANSYS Confidential information ***

c         *** Copyright ANSYS.  All Rights Reserved.
c         *** ansys, inc.

c  input arguments:
c       elem   (int,sc,in)         - element label (number)
c       ielc   (int,ar(IELCSZ),in) - array of element type characteristics
c                                     (IELCSZ = array size, defined in echprm)
c       elmdat (int,ar(EL_DIM),in) - array of element data
c       eomaskL(LONG,sc,in)        - bit pattern for element output
c                                    (see outpcm)
c       nodes  (int,ar(nnod),in)   - array of element node numbers
c                                   (nnod = number of nodes; 1 in this case)
c       locsvrL(LONG,sc,in)        - location of the saved variables
c                                              on file .esav for this element
c       kelreq (int,ar(10),in)     - matrix and load vector form requests
c                                    (indices for kelreq are given with output
c                                                          arguments below)
c       kelfil (int,ar(10),in)     - keys indicating incoming matrices and
c                                    load vectors (indices for kelfil are the
c                                    same as given for kelreq with output
c                                    arguments below)
c       nr     (int,sc,in)         - matrix and load vector size
c       xyz    (dp,ar(6,nnod),in)  - nodal coordinates (orig) and rotation angle
c       u      (dp,ar(nr,5),in)    - element nodal solution values

c  output arguments:
c       kelout (int,ar(10),out)    - keys indicating created matrices and
c                                         load vectors (indices for kelout
c                                            are the same as for kelreq below,
c                                            as well as kelin and kelout later)
c       zs     (dp,ar(nr,nr),inout)- stiffness/conductivity matrix (kelreq(1))
c       zass   (dp,ar(nr,nr),inout)- mass matrix                   (kelreq(2))
c       damp   (dp,ar(nr,nr),inout)- damping/specific heat matrix  (kelreq(3))
```

```
c       gstif  (dp,ar(nr,nr),inout)- stress stiffness matrix       (kelreq(4))
c       zsc    (dp,ar(nr),out)     - applied f vector              (kelreq(5))
c       zscnr  (dp,ar(nr),out)     - n-r restoring f vector        (kelreq(6))
c                                    or imaginary f vector         (kelreq(7))
c       elvol  (dp,sc,out)         - element volume
c       elmass (dp,sc,out)         - element mass
c       center (dp,ar(3),out)      - centroid location
c       elener (dp,ar(5),out)      - element energies
c       edindxL(LONG,ar(*),out)   - element result data file indexes
c       lcerstL(LONG,sc,inout)     - position on result file
```

2.1.3.4.1. Subroutines uel101 through uel105

The input and output arguments for subroutines `uel101`, `uel102`, `uel103`, `uel104`, and `uel105`are identical to subroutine `uel100` listed above.

2.1.3.5. Subroutine uep100 (Printing Output for User Elements in POST1 via PRESOL,ELEM)

```
*deck,uep100                      USERDISTRIB
      subroutine uep100 (iott,elem,nodes,mat, kept,tem,
     x  kemn,fluen, kems,force, kens,sig, keel,epel,
     x  keth,eptho,epswel,epino, kenl,sigepl,sigrat,hpres,epeq,
     x  kepl,eppl, kecr,epcrp)
c
c *** primary function:    produce printed output for user100
c
c         *** Copyright ANSYS.  All Rights Reserved.
c         *** ansys, inc.
c *** Notice - This file contains ANSYS Confidential information ***
c
c  **********  this subroutine is provided for user information  *********
c *** user programmable features may not be used in parallel processing ***
c
c  input arguments:
c     iott     (int,sc,in)         - output unit number
c     elem     (int,sc,in)         - element number
c     nodes    (int,ar(2),in)      - node numbers
c     mat      (int,sc,in)         - material number
c     kept     (int,sc,in)         - key to print temperatures
c     tem      (dp,ar(2),in)       - nodal temperatures
c     kemn     (inr,sc,in)         - key to print fluences
c     fluen    (dp,ar(2),in)       - neutron fluences
c     kems     (int,sc,in)         - key to print moment forces
c     force    (int,sc,in)         - member force fx
c     kens     (int,sc,in)         - key to print strains
c     sig      (dp,sc,in)          - axial stress
c     keel     (int,sc,in)         - key to print elastic strain
c     epel     (dp,sc,in)          - axial elastic strain
c     keth     (int,sc,in)         - key to print thermal,initial,swelling strai
c     eptho    (dp,sc,in)          - axial thermal strain
c     epswel   (dp,sc,in)          - swelling strain
c     epino    (dp,sc,in)          - initial axial strain
c     kenl     (int,sc,in)         - key set if any nonlinear materials present
c     sigepl   (dp,sc,in)          - stress in stress-strain curve
c     sigrat   (dp,sc,in)          - stress ratio
c     hpres    (dp,sc,in)          - hydrostatic pressure
c     epeq     (dp,sc,in)          - plastic equivalent strain
c     kepl     (int,sc,in)         - key to print plastic strain
c     eppl     (dp,sc,in)          - plastic strain
c     kecr     (int,sc,in)         - key to print creep strain
c     epcrp    (dp,sc,in)          - creep strain
c
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c  output arguments:     none
c
```

2.1.3.5.1. Subroutines uep101 through uep105

The source code for subroutines `uep101`, `uep102`, `uep103`, `uep104`, and `uep105` is identical to subroutine `uep100` listed above.

2.1.3.6. Subroutine usertr (Adjusting the Nodal Orientation Matrix)

```
*deck,usertr                      USERDISTRIB
      subroutine usertr (node,tr)
c *** primary function:  adjust nodal orientation matrix
c     secondary function: study nodal orientation matrix
c       accessed with ielc(notran) = -100
c
c         *** Copyright ANSYS.  All Rights Reserved.
c         *** ansys, inc.
c
c *** Notice - This file contains ANSYS Confidential information ***
c
c     typ=int,dp,log,chr,dcp   siz=sc,ar(n)   intent=in,out,inout
c
c  input arguments:
c     variable (typ,siz,intent)    description
c     node     (int,sc,in)       - node number being acted upon
c     tr       (dp,ar(32,32),inout) - nodal to global orientation matrix
c
c  output arguments:
c     variable (typ,siz,intent)    description
c     tr       (dp,ar(32,32),inout) - nodal to global orientation matrix
c
c          tr is a matrix that is already defined based on the degrees
c          of freedom selected.
c          it does not normally need to be changed.
c          it may be printed out here to study.  its functional size is
c          nr by nr, where nr is the number of degrees of freedom in the
c          element
c
```

2.1.3.7. Subroutine userac (Accessing Element Information)

This subroutine is provided for demonstration purposes.

```
*deck,userac                      USERDISTRIB
      subroutine userac (elem)
c *** primary function:  To demonstrate user access of element information.
c --- Given the element number, all information about the element is avaiable.
c --- Starting with elmdat, one can get the element type, real constant number,
c --- the material number, the element coordinate system number, as well as
c --- the node numbers.  Then, one can get more information about any or all
c --- of these things.  The below demonstrates getting the element type and
c --- real constants.
c
c         *** Copyright ANSYS.  All Rights Reserved.
c         *** ansys, inc.
c *** Notice - This file contains ANSYS Confidential information ***
c
c  input arguments:
c     variable (typ,siz,intent)    description
c     elem     (int,sc,in)       - element number
```

```
c
c  output arguments:
c     none
c
```

2.2. Supporting Subroutines for Element Creation

The following subroutines support the user subroutines for creating a new element (p. 151) (using the database-access method (p. 171)):

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.2.1. Subroutine nminfo (Returning Element Reference Names)

```
*deck,nminfo
      subroutine nminfo (ielc,rname)
c *** primary function:    set element reference names
c *** secondary functions: none
c       -------- to get name back, use nameiq
c
c *** Notice - This file contains ANSYS Confidential information ***
c
c  input arguments:
c     variable (typ,siz,intent)      description
c     ielc     (int,ar(*),inout)   - element characteristic vector
c     rname    (chr,sc,in)         - 8 character reference name
c
c  output arguments:
c     variable (typ,siz,intent)      description
c     ielc     (int,ar(*),inout)   - element characteristic vector with
c                                      element name encoded
c
```

2.2.2. Subroutine svgidx (Fetching the Index for Saved Variables)

```
*deck,svgidx
      subroutine svgidx (locsvr,svindx)
c *** primary function:    get the index for saved variables

c *** Notice - This file contains ANSYS Confidential information ***

c  input arguments:
c     locsvr   (LONGINT,sc,in)   - pointer to location of index

c  output arguments:
c     svindx   (int,ar(20),out) - the 20 word index of svr variables
c                                  1,2-starting loc of this eles svr sets
c                                  3- length of eles svr sets
c                                  4-11-relative starting loc for each set
c                                     4-structural svrs
c                                     5-thermal/electric/fluid svrs
c                                     6-magnetic svrs
c                                     7-nonlinear svrs
c                                     8-plasticity svrs
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                     9-creep svrs
c                                    10-coupled svrs
c                                    11-user svrs
c                                    12-initial strain svrs
c                                    13-section data after FiberSIM conversion
c                                                          (shell181 only)
c                                    14-20 spares
```

2.2.3. Subroutine svrget (Fetching Saved Variable Data for an Element)

```
*deck,svrget
      subroutine svrget (svindx,nset,nsvr,svr,svrlen)
c *** primary function:    get svr data set for an element

c *** Notice - This file contains ANSYS Confidential information ***

c  input arguments:
c     svindx   (int,ar(20),in)    - index for svr for this element (see svgidx)
c     nset     (int,sc,in)        - the set number in this index
c                                   =  1 - structural svrs
c                                   =  2 - thermal/electric/fluid svrs
c                                   =  3 - magnetic svrs
c                                   =  4 - nonlinear svrs
c                                   =  5 - plasticity svrs
c                                   =  6 - creep svrs
c                                   =  7 - coupled svrs
c                                   =  8 - user svrs
c                                   =  9 - initial stress svrs
c                                             (2,42,82,45,92,95 only)
c                                   = 10 - section data after FiberSIM conversion
c                                                          (shell181 only)
c                                   = 11-17 - spares (note that the first three
c                                              items in svindx are not available)
c     nsvr     (int,sc,inout)     - number of dp words expected in this set
c     svrlen   (int,sc,in)        - len of svr array

c  output arguments:
c     nsvr     (int,sc,inout)     - number of dp words in this set
c     svr      (dp,ar(svrlen),in) - data in this set
```

2.2.4. Subroutine svrput (Writing an Element's Saved Variable Set)

```
*deck,svrput
      subroutine svrput (svindx,nset,leng,svr)
c *** primary function:    write out a svr data set for an element

c *** Notice - This file contains ANSYS Confidential information ***

c  input arguments:
c     svindx   (int,ar(20),inout)- the index for svr for this element
c                                                              (see svgidx)
c     nset     (int,sc,in)       - the set number in this index (same as svrget)
c                                   =  1 - structural svrs
c                                   =  2 - thermal/electric/fluid svrs
c                                   =  3 - magnetic svrs
c                                   =  4 - nonlinear svrs
c                                   =  5 - plasticity svrs
c                                   =  6 - creep svrs
c                                   =  7 - coupled svrs
c                                   =  8 - user svrs
c                                   =  9 - initial stress svrs
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                                 (2,42,82,45,92,95 only)
c                                  = 10 - section data after FiberSIM conversion
c                                                              (shell181 only)
c                                  = 11-17 - spares (note that the first three
c                                            items in svindx are not available)
c     leng     (int,sc,in)       - number of dp words in this set
c     svr      (dp,ar(leng),in)  - data in this set

c  output arguments:
c     svindx   (int,ar(10,2),inout)- updated index
```

2.2.5. Subroutine svpidx (Writing the Saved Variable Element Index to a File)

```
*deck,svpidx
      subroutine svpidx (locsvr,svindx)
c *** primary function:    write the svr element index onto file
c *** secondary functions: update the locsvr pointer to next element

c *** Notice - This file contains ANSYS Confidential information ***

c  input arguments:
c     locsvr   (LONGINT,sc,inout)    - pointer to start of svr for element
c     svindx   (int,ar(10,2),in)     - index to svr for this element
c                                        low and high parts of 64 bit address

c  output arguments:
c     locsvr   (LONGINT,sc,inout)    - pointer to start of svr for next element
```

2.2.6. Subroutine mreuse (Determining Which Element Matrices Can Be Reused)

```
*deck,mreuse
      subroutine mreuse (kelrqq,kelfil,elem,ielc,kmasrt,knlmg,kconve,
     x kpheno,kprop,nprop,prop,propo,krvro,rvr,rvro,amodo,asymo, kelin)
c *** primary function:
c         determine which Matrices can be REUSEd and which must be recomputed
c          from iteration to iteration.
c          Note:  a few special elements have some supplementary logic
c          to adjust these results further.  No attempt as been made to
c          include all such logic in these routines.
c
c          Second note:  this logic is essentially the same as the old
c          sfrm logic.  Hopefully, further simplifications and enhancements
c          will be made in the future. (Especially in gap elements and in
c          multilayer elements)
c          the whole idea of kpheno, a holdover from the sfrm routines,
c          needs to be looked at and possibly eliminated.
c
c *** Notice - This file contains ANSYS Confidential information ***
c
c  input arguments:
c     kelrqq   (int,ar(10),in)     - request keys (needed for this analysis)
c     kelfil   (int,ar(10),in)     - keys indicating matrices on the file
c     elem     (int,sc,in)         - element number
c     ielc     (int,ar(IELCSZ),in) - array of element type characteristics
c     kmasrt   (int,sc,in)         - does the mass matrix have rotational DOF?
c                                       0 - no     1 - yes(with nlgeom, sfrm1n)
c     knlmg    (int,sc,in)         - nonlinear magnetic curve exists in this
c                                     element
c                                      0 - no     1 - yes
```

```
c     kconve   (int,sc,in)         - key indicating existence of convections
c                                     in this element
c                                      0,1 - no     2 or more - yes
c                                     must be input as 'i' if not used, as is
c                                     changed in this routine(for analyzer).
c                                     i = 0 must be used in calling routine
c                                     if kpheno = 1.
c     kpheno   (int,sc,in)         - key for type of phenomenon/level of check
c                                     0 - structural like old sfrm1n,1s,3n,3s,fl
c                                     1 - thermal  like old sfrm1c,1t,2t,3t
c                                     2 - electrical/magnetic like some of old
c                                                                    sfrmpo
c                                     3 - general    like old sfrmoo
c     kprop    (int,sc,in)         - key indicating which material properties
c                                     in the prop vector that need to be
c                                     checked (see below)
c     nprop    (int,sc,in)         - number of properties
c     prop     (dp,ar(nprop),in)   - current mat props
c     propo    (dp,ar(nprop),inout)- previous material properties
c     krvro    (int,sc,in)         -
c            = 0 - real constants are used by this element, and the old
c                  values(rvro) have been saved; or the element does not
c                  use real constants.  Any change of real constants
c                  causes all matrices to be reformed.
c            = 1 - real constants are used by this element and the old
c                  values(rvro) have been saved.  However, any change
c                  of real constants will cause the run to terminate,
c                  because the results would be too unpredictable.
c                  (e.g. gap elements)
c            = 2 - element is nonlinear, so do not bother to check
c            = 3 - real constants are used by this element, and the old
c                  values(rvro) have been saved.  However, no checking is
c                  done in this routine because of needed customized logic.
c            = 4 - real constants are used by this element, but the old
c                  values(rvro) have not been saved because it was
c                  decided not to use this much storage.  therefore, no check
c                  can be made to determine if matrices should be reformed.
c                  (e.g. 100 layer elements)
c            = 5 - real constants are used by this element, but the old
c                  values(rvro) have not been saved because the real
c                  constants have no effect on matrix formulation.
c                  (e.g. acoustic elements)
c     rvr      (dp,ar(*),in)       - current real constants
c     rvro     (dp,ar(*),inout)    - previous real constants
c     amodo    (dp,sc,inout)       - previous value of mode
c     asymo    (dp,sc,inout)       - previous value of isym
c
c  output arguments:
c     propo    (dp,ar(nprop),inout)- current material properties
c     rvro     (dp,ar(*),inout)    - current real constants
c     amodo    (dp,sc,inout)       - current value of mode
c     asymo    (dp,sc,inout)       - current value of isym
c     kelin    (int,ar(10),out)    - keys indicating matrices to form
c
```

2.2.7. Subroutine subrd (Reading Element Load Data for a Substructure Generation Run)

```
*deck,subrd
      subroutine subrd (iel,key,nd,vect,ka)
c *** primary function:    read element load data from file for substructure
c                          generation run
c *** secondary functions: none

c *** Notice - This file contains ANSYS Confidential information ***
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

```
c  input arguments:
c     iel      (int,sc,in)       - element number
c     key      (int,sc,in)       - type of load data
c                                 = 1 temperature
c                                 = 2 fluences
c                                 = 3 heat generation rates
c                                 = 4 current densities
c                                 = 9 end pressures (needed for beams/pipes)
c                                 =10 pressures
c                                 =11 film coefficients
c                                 =12 bulk temperatures
c                                 =13 extra displacement shapes
c                                 =14 thermal strains(eptho in el42)
c                                 =15 thermal flux (as in el55)
c                                 =16 initial strains(epino in el01)
c                                 =17 magnetic virtual displacements
c                                 =18 calculated source field(hsn in el96)
c                                 =20 element load vector
c                                 =30 copy - do not scale(tempev in el42)
c                                     first load step only
c     nd       (int,sc,in)       - number of data items

c  output arguments:
c     vect     (dp,ar(nd),out)   - array of load data
c     ka       (int,sc,out)      - load activation key
c                                  = 0 no load for this data
c                                  = 1 load is active
```

2.2.8. Subroutine subwrt (Writing an Element Load Vector to a File for a Substructure Generation Run)

```
*deck,subwrt
      subroutine subwrt (iel,nvect,kkey,nd,vect,ref)
c *** primary function:    write element load vect to file for substructure
c                          generation run
c *** secondary functions: none

c *** Notice - This file contains ANSYS Confidential information ***

c  input arguments:
c     iel      (int,sc,in)       - element number
c     nvect    (int,sc,in)       - number of load vectors
c                                   (current load step number)
c     kkey     (int,sc,in)       - type of load vect
c                                 = 1 temperature
c                                 = 2 fluences
c                                 = 3 heat generation rates
c                                 = 4 current densities
c                                 = 9 end pressures
c                                 =10 pressures
c                                 =11 film coefficients
c                                 =12 bulk temperatures
c                                 =13 extra displacement shapes
c                                 =14 thermal strains(eptho in el42)
c                                 =15 thermal flux (as in el55)
c                                 =16 initial strains(epino in el01)
c                                 =17 magnetic virtual displacements
c                                 =18 calculated source field(hsn in el96)
c                                 =20 element load vector
c                                 =30 copy - do not scale(tempev in el42)
c     nd       (int,sc,in)       - number of vect items
c     vect     (dp,ar(nd),in)    - array of load data
c     ref      (dp,sc,in)        - reference value for zero load

c  output arguments: none
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.2.9. Subroutine rvrget (Fetching Real Constants for an Element)

```
*deck,rvrget
      subroutine rvrget (iel,ireal,ielc,nrvr,rvr)
c *** primary function:  get the real constants for an element

c    typ=int,dp,log,chr,dcp   siz=sc,ar(n),func    intent=in,out,inout

c     variable (typ,siz,intent)     description
c        iel      (int,sc,in)       - element number
c        ireal    (int,sc,in)       - real constant set number
c        ielc     (int,ar(*),in)    - elment type characteristics

c     output arguments:
c        nrvr     (int,sc,out)      - number of real variables
c        rvr      (dp,ar(*),out)    - element real constants

c *** mpg magnetic element usage - iel ?
```

2.2.10. Subroutine propev (Evaluating a Group of Material Properties)

```
*deck,propev
      subroutine propev (iel,mtr,lp,tem,prop,n)
c *** primary function:   to evaluate a group of material properties

c     propev is used to pass two or more material property numbers
c     thru the lp array to determine which temperature dependent
c     material properties are to be evaluated.
c     thus, the 3 prope1 calls:

c        call prope1 (elem,mat, 1,tem,e(1))
c        call prope1 (elem,mat,10,tem,alpha)
c        call prope1 (elem,mat,13,tem,dens)

c     should be combined as:

c        integer lp(3)
c        data lp /1,10,13/
c        call propev (elem,mat,lp(1),tem,prop(1),3)

c *** Notice - This file contains ANSYS Confidential information ***

c  input arguments:
c     iel  (int,sc,in)       - element number
c     mtr  (int,sc,in)       - material number(input quantity mat, mat comma
c     lp   (int,ar(n),in)    - keys for which specific value is requested
c                                     each group must be in ascending
c                                     order  (ex,ey,ez, etc)
c                                  if negative, a required property
c                                  if zero, leave prop term unchanged
c        ---- MP command labels --------
c        EX  = 1, EY  = 2, EZ  = 3, NUXY= 4, NUYZ= 5, NUXZ= 6, GXY = 7, GYZ = 8,
c        GXZ = 9, ALPX=10, ALPY=11, ALPZ=12, DENS=13, MU  =14, DAMP=15, KXX =16,
c        KYY =17, KZZ =18, RSVX=19, RSVY=20, RSVZ=21, C   =22, HF  =23, VISC=24,
c        EMIS=25, ENTH=26, LSST=27, PRXY=28, PRYZ=29, PRXZ=30, MURX=31, MURY=32,
c        MURZ=33, PERX=34, PERY=35, PERZ=36, MGXX=37, MGYY=38, MGZZ=39, EGXX=40,
c        EGYY=41, EGZZ=42, SBKX=43, SBKY=44, SBKZ=45, SONC=46, DMPS=47, ELIM=48,
c        USR1=49, USR2=50, USR3=51, USR4=51, FLUI=53, ORTH=54, CABL=55, RIGI=56,
c        HGLS=57, BVIS=58, QRAT=59, REFT=60, CTEX=61, CTEY=62, CTEZ=63, THSX=64,
c        THSY=65, THSZ=66, DMPR=67, LSSM=68, BETD=69, ALPD=70, RH  =71, DXX =72,
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c        DYY =73, DZZ =74, BETX=75, BETY=76, BETZ=77, CSAT=78, CREF=79, CVH =80,
c        UMID=81, UVID=82
c                                              (see mpinit for uncommented code)
c      (see chapter 2 of the elements volume of the user's manual
c       for a detailed description))

c     tem      (dp,sc,in)        - temperature at which to evaluate material
c     n        (int,sc,in)       - number of properties to be evaluated.
c                                   (20 maximum)
c                                   If n = 1, use prope1 instead.

c  output arguments:
c     prop     (dp,ar(n),out)    - values of material property
```

2.2.11. Subroutine prope1 (Evaluating One Material Property)

```
*deck,prope1
      subroutine prope1 (iel,mtr,icon,tem,prop1)
c *** primary function:   to evaluate one material property
c                           (if multiple material properties are to
c                            be evaluated, use propev)
c *** secondary functions: to ensure that certain required props are present

c *** Notice - This file contains ANSYS Confidential information ***

c  input arguments:
c     iel      (int,sc,in)       - element number
c     mtr      (int,sc,in)       - material number
c     icon     (int,sc,in)       - key for which specific value is requested
c                                  (negative if property is required)
c        ---- MP command labels --------
c        EX  = 1, EY  = 2, EZ  = 3, NUXY= 4, NUYZ= 5, NUXZ= 6, GXY = 7, GYZ = 8,
c        GXZ = 9, ALPX=10, ALPY=11, ALPZ=12, DENS=13, MU  =14, DAMP=15, KXX =16,
c        KYY =17, KZZ =18, RSVX=19, RSVY=20, RSVZ=21, C   =22, HF  =23, VISC=24,
c        EMIS=25, ENTH=26, LSST=27, PRXY=28, PRYZ=29, PRXZ=30, MURX=31, MURY=32,
c        MURZ=33, PERX=34, PERY=35, PERZ=36, MGXX=37, MGYY=38, MGZZ=39, EGXX=40,
c        EGYY=41, EGZZ=42, SBKX=43, SBKY=44, SBKZ=45, SONC=46, DMPS=47, ELIM=48,
c        USR1=49, USR2=50, USR3=51, USR4=51, FLUI=53, ORTH=54, CABL=55, RIGI=56,
c        HGLS=57, BVIS=58, QRAT=59, REFT=60, CTEX=61, CTEY=62, CTEZ=63, THSX=64,
c        THSY=65, THSZ=66, DMPR=67, LSSM=68, BETD=69, ALPD=70, RH  =71, DXX =72,
c        DYY =73, DZZ =74, BETX=75, BETY=76, BETZ=77, CSAT=78, CREF=79, CVH =80,
c        UMID=81, UVID=82
c                                              (see mpinit for uncommented code)

c     tem      (dp,sc,in)        - temperature at which to evaluate material

c  output arguments:
c     prop1    (dp,sc,out)       - value of material property
```

2.2.12. Subroutine pstev1 (Evaluating Material Properties for 1-D Elements)

```
*deck,pstev1
      subroutine pstev1 (elem,matin,tem,prop)
c *** primary function:    to evaluate material properites for 1-d elements
c
c *** Notice - This file contains ANSYS Confidential information ***
c
c  input arguments:
c     elem     (int,sc,in)         - element number (for anserr)
c     matin    (int,sc,in)         - material reference number
c                                    if negative, no required properties
```

```
c     tem      (dp,sc,in)          - temperature for evaluation
c
c  output arguments:
c     prop     (dp,ar(5),out)      - material properties: ex,nuxy,gxy,alpx,dens
c
```

2.2.13. Subroutine tbuser (Retrieving User Table Data)

```
*deck,tbuser
      subroutine tbuser (mat,numitm,tbprop)
c *** primary function:    return the tb data for the user table

c *** Notice - This file contains ANSYS Confidential information ***

c  input arguments:
c     mat      (int,sc,in)         - material property number
c     numitm   (int,sc,in)         - the number of data items requested

c  output arguments:
c     tbprop   (dp,ar(numitm),out) - array of tb data
```

2.2.14. Subroutine plast1 (Updating an Element's Plastic History)

```
*deck,plast1
      subroutine plast1 (option,elem,intpt,mat,kstartL,tem,dtem,e,
     x                   ktform,dens,flu,dflu,epel,eppl,statev,usvr,
     x                   epeq,plwork,sigepl,sigrat,et)
c *** primary function:    to update the plastic history (for 1 component)
c                          used by:  LINK1, LINK8, BEAM23, BEAM24, and
c                                    SOLID65(reinforcing)
c *** secondary functions: to compute the material tangent matrix if requested

c *** Notice - This file contains ANSYS Confidential information ***

c  input arguments:
c     option   (int,sc,in)         - plasticity option
c     elem     (int,sc,in)         - element number (label)
c     intpt    (int,sc,in)         - element integration point number
c     mat      (int,sc,in)         - material reference number
c     kstartL  (intL,sc,in)        - virtual starting address of the data table
c     tem      (dp,sc,in)          - temperature at the end of this substep
c     dtem     (dp,sc,in)          - temperature increment over this substep
c     e        (dp,sc,in)          - elastic modulus
c     ktform   (int,sc,in)         - request key for tangent matrix formation
c     dens     (dp,sc,in)          - material density
c     flu      (dp,sc,in)          - fluence at the end of this substep
c     dflu     (dp,sc,in)          - fluence increment over this substep
c     epel     (dp,sc,inout)       - modified total strain (trial strain)
c     eppl     (dp,sc,inout)       - plastic strain at previous substep
c     statev   (dp,ar(6),inout)    - state variables at previous substep
c     usvr     (dp,ar(*),inout)    - user-defined state variables (for userpl)
c     epeq     (dp,sc,inout)       - effective plastic strain at prev substep
c     plwork   (dp,sc,inout)       - accumulated plastic work at prev substep

c  output arguments:
c     epel     (dp,sc,inout)       - elastic strain
c     eppl     (dp,sc,inout)       - updated plastic strain
c     statev   (dp,ar(6),inout)    - updated state variables
c     usvr     (dp,ar(*),inout)    - updated user-defined state variables
c     epeq     (dp,sc,inout)       - updated effective plastic strain
c     plwork   (dp,sc,inout)       - updated accumulated plastic work
c     sigepl   (dp,sc,out)         - stress value on stress-strain curve
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

```
c     sigrat   (dp,sc,out)         - ratio of trial stress to yield stress
c     et       (dp,sc,out)         - tangent modulus

c  internal variables:
c     deppl    (dp,sc)             - equivalent plastic strain increment
```

2.2.15. Subroutine plast3 (Updating an Element's Plastic History, 4 or 6 components)

```
*deck,plast3
      subroutine plast3 (option,elem,intpt,mat,kstartL,ncomp,tem,dtem,
     x prop,d,ktform,dens,flu,dflu,epel,eppl,statev,usvr,epeq,plwork,
     x sigepl,sigrat,dt,kplst,dtt,cmel)
c *** primary function:    to update the plastic history (for 4 or 6 components)
c       used by:  PLANE13, PIPE20, SHELL43, PIPE60,
c                 SOLID62, SOLID65, SHELL91, SHELL93, SOLID191
c       and by way of plast3creep : PLANE42, SOLID45, PLANE82, SOLID92, SOLID95

c *** secondary functions: to compute the material tangent matrix if requested

c *** Notice - This file contains ANSYS Confidential information ***

c  input arguments:
c     option   (int,sc,in)         - plasticity option
c     elem     (int,sc,in)         - element number (label)
c     intpt    (int,sc,in)         - element integration point number
c     mat      (int,sc,in)         - material reference number
c     kstartL  (intL,sc,in)        - virtual starting address of the data table
c     ncomp    (int,sc,in)         - number of stress/strain components (4 or 6)
c     tem      (dp,sc,in)          - temperature at the end of this substep
c     dtem     (dp,sc,in)          - temperature increment over this substep
c     prop     (dp,ar(9),in)       - material property array (ex,ey,ez,
c                                    gxy,gyz,gxz, uxy,uyz,uxz)
c     d        (dp,ar(ncomp,ncomp),in) - elastic stress-strain matrix
c     ktform   (int,sc,in)         - request key for tangent matrix formation
c     dens     (dp,sc,in)          - material density
c     flu      (dp,sc,in)          - fluence at the end of this substep
c     dflu     (dp,sc,in)          - fluence increment over this substep
c     epel     (dp,ar(ncomp),inout)- modified total strain (trial strain)
c     eppl     (dp,ar(ncomp),inout)- plastic strain at previous substep
c     statev   (dp,ar(ncomp,6),inout)- state variables at previous substep
c     usvr     (dp,ar(*),inout)    - user-defined state variables (for pluser)
c     epeq     (dp,sc,inout)       - effective plastic strain at prev substep
c     plwork   (dp,sc,inout)       - accumulated plastic work at prev substep
c     kplst    (int,sc,in)         - plane stress key (form dtt if kplst=1)

c  output arguments:
c     epel     (dp,ar(ncomp),inout)- elastic strain
c     eppl     (dp,ar(ncomp),inout)- updated plastic strain
c     statev   (dp,ar(ncomp,6),inout)- updated state variables
c     usvr     (dp,ar(*),inout)    - updated user-defined state variables
c     epeq     (dp,sc,inout)       - updated effective plastic strain
c     plwork   (dp,sc,inout)       - updated accumulated plastic work
c     sigepl   (dp,sc,out)         - stress value on stress-strain curve
c     sigrat   (dp,sc,out)         - ratio of trial stress to yield stress
c     dt       (dp,ar(ncomp,ncomp),out)- material modulus modified by dscpar
c     dtt      (dp,ar(ncomp,ncomp),out)- consistent tangent modulus
c                                       (formed only if kplst=1)

c  internal variables:
c     deppl    (dp,sc)             - equivalent plastic strain increment
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.2.16. Subroutine creep1 (Updating an Element's Creep History)

```
*deck,creep1
      subroutine creep1 (option,elem,intpt,mat,kstartL,epel,e,epcrp,
     x statev,usvr,tem,dtem,fluen,dflu,sig)
c *** primary function:    to update the creep history for 1-d elements
c                          used by:  LINK1, LINK8, BEAM23, BEAM24, and
c                                    SOLID65(reinforcing)

c *** Notice - This file contains ANSYS Confidential information ***

c  input arguments:
c     option    (int,sc,in)        - creep option
c     elem     (int,sc,in)         - element number (label)
c     intpt    (int,sc,in)         - element integration point number
c     mat      (int,sc,in)         - material reference number
c     kstartL  (intL,sc,in)        - virtual starting address of the data table
c     epel     (dp,sc,inout)       - elastic strain
c     e        (dp,sc,in)          - elastic modulus
c     epcrp    (dp,sc,inout)       - creep strain at previous substep
c     statev   (dp,ar(7),inout)    - state variables at previous substep
c     usvr     (dp,ar(*),inout)    - user-defined state variables (for usercr)
c     tem      (dp,sc,in)          - temperature at the end of this substep
c     dtem     (dp,sc,in)          - temperature increment over this substep
c     fluen    (dp,sc,in)          - fluence at the end of this substep
c     dflu     (dp,sc,in)          - fluence increment over this substep
c     epel     (dp,sc,inout)       - elastic strain adjusted for creep increment
c     sig      (dp,sc,inout)       - stress (not really used)

c  output arguments:
c     epcrp    (dp,sc,inout)       - updated creep strain
c     statev   (dp,ar(7),inout)    - updated state variables
c     usvr     (dp,ar(*),inout)    - updated user-defined state variables
c     sig      (dp,sc,inout)       - stress (recomputed if requested)
```

2.2.17. Subroutine creep3 (Updating an Element's Creep History, 3-D Elements)

```
*deck,creep3
      subroutine creep3 (option,elem,intpt,mat,kstartL,ncomp,epel,e,
     x posn,d,epcrp,statev,usvr,tem,dtem,fluen,dflu,kplst,sig,hsig)
c *** primary function:    to update the creep history for 3-d elements
c       used by:  PLANE13, PIPE20, PLANE42, SHELL43, SOLID45,
c                 PIPE60, SOLID62, SOLID65, PLANE82, SHELL91,
c                 SOLID92, SHELL93, SOLID95, SOLID191

c *** Notice - This file contains ANSYS Confidential information ***

c  input arguments:
c     option   (int,sc,in)         - creep option
c     elem     (int,sc,in)         - element number (label)
c     intpt    (int,sc,in)         - element integration point number
c     mat      (int,sc,in)         - material reference number
c     kstartL  (intL,sc,in)        - virtual starting address of the data table
c     ncomp    (int,sc,in)         - number of stress/strain components (4 or 6)
c     epel     (dp,ar(ncomp),inout)- elastic strain
c     e        (dp,sc,in)          - elastic young'S MODULUS
c     posn     (dp,sc,in)          - poisson'S RATIO
c     d        (dp,ar(ncomp,ncomp),in) - elastic stress-strain matrix
c     epcrp    (dp,ar(ncomp),inout)- creep strain at previous substep
c     statev   (dp,ar(ncomp*5+2),inout)- state variables at previous substep
c     usvr     (dp,ar(*),inout)    - user-defined state variables (for usercr)
c     tem      (dp,sc,in)          - temperature at the end of this substep
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c     dtem     (dp,sc,in)          - temperature increment over this substep
c     fluen    (dp,sc,in)          - fluence at the end of this substep
c     dflu     (dp,sc,in)          - fluence increment over this substep
c     kplst    (int,sc,in)         - plane stress/plane strain key
c     sig      (dp,ar(ncomp),inout)- stresses (not used in input)
c     hsig     (dp,ar(1),inout)    - hydrostatic stress (not used in input)

c  output arguments:
c     epel     (dp,ar(ncomp),inout)- elastic strain adjusted for creep increment
c     epcrp    (dp,ar(ncomp),inout)- updated creep strain
c     statev   (dp,ar(ncomp*5+2),inout)- updated state variables
c     usvr     (dp,ar(*),inout)    - updated user-defined state variables
c     sig      (dp,ar(ncomp),inout)- stresses (redefined if c13 > 0)
c     hsig     (dp,sc,inout)       - hydrostatic stress (redefined if c13 > 0)
```

2.2.18. Subroutine swell1 (Updating an Element's Swelling History)

```
*deck,swell1
      subroutine swell1 (option,elem,intpt,mat,kstartL,epswel,epel,e,
     x fluen,dfluen,tem,dtem,usvr)
c *** primary function:    to update the swelling history for 1-d elements
c                          used by:  LINK1, LINK8, BEAM23, and BEAM24

c *** Notice - This file contains ANSYS Confidential information ***

c  input arguments:
c     option   (int,sc,in)         - swelling option
c     elem     (int,sc,in)         - element number (label)
c     intpt    (int,sc,in)         - element integration point number
c     mat      (int,sc,in)         - material reference number
c     kstartL  (intL,sc,in)        - virtual starting address of the data table
c     epswel   (dp,sc,inout)       - swell strain at previous substep
c     epel     (dp,sc,inout)       - elastic strain
c     e        (dp,sc,in)          - elastic young'S MODULUS
c     fluen    (dp,sc,in)          - fluence at the end of this substep
c     dfluen   (dp,sc,in)          - fluence increment over this substep
c     tem      (dp,sc,in)          - temperature at the end of this substep
c     dtem     (dp,sc,in)          - temperature increment over this substep
c     usvr     (dp,ar(*),inout)    - user-defined state variables (for usersw)

c  output arguments:
c     epel     (dp,sc,inout)       - elastic strain adjusted for swelling inc
c     epswel   (dp,sc,inout)       - updated swelling strain
c     usvr     (dp,ar(*),inout)    - updated user-defined state variables
```

2.2.19. Subroutine swell3 (Updating an Element's Swelling History, 3-D Elements)

```
*deck,swell3
      subroutine swell3 (option,elem,intpt,mat,kstartL,ncomp,epswel,
     x                   epel,e,nuxy,fluen,dfluen,tem,dtem,usvr)
c *** primary function:    to update the swelling history for 3-d elements
c       used by:  PLANE13, PIPE20, PLANE42, SHELL43, SOLID45,
c                 PIPE60, SOLID62, PLANE82, SHELL91, SOLID92,
c                 SHELL93, SOLID95, SOLID191

c *** Notice - This file contains ANSYS Confidential information ***

c  input arguments:
c     option   (int,sc,in)         - swelling option
```

```
c     elem     (int,sc,in)         - element number (label)
c     intpt    (int,sc,in)         - element integration point number
c     mat      (int,sc,in)         - material reference number
c     kstartL  (intL,sc,in)        - virtual starting address of the data table
c     ncomp    (int,sc,in)         - number of stress/strain components (4 or 6)
c     epswel   (dp,sc,inout)       - swell strain at previous substep
c     epel     (dp,ar(ncomp),inout)- elastic strain
c     e        (dp,sc,in)          - elastic young'S MODULUS
c     nuxy     (dp,sc,in)          - poisson'S RATIO
c     fluen    (dp,sc,in)          - fluence at the end of this substep
c     dfluen   (dp,sc,in)          - fluence increment over this substep
c     tem      (dp,sc,in)          - temperature at the end of this substep
c     dtem     (dp,sc,in)          - temperature increment over this substep
c     usvr     (dp,ar(*),inout)    - user-defined state variables (for usersw)

c  output arguments:
c     epel     (dp,ar(ncomp),inout)- elastic strain adjusted for swelling inc
c     epswel   (dp,sc,inout)       - updated swelling strain
c     usvr     (dp,ar(*),inout)    - updated user-defined state variables
```

2.2.20. Function elLenPsvrBuf (Determining Additional ESAV Record for Plasticity)

```
*deck,elLenPsvrBuf
      function elLenPsvrBuf (mat, plOpt, ncomp)

c**********************************************************************

c     *** primary function:
c              determine additional esave record for plasticity

c     input arguments
c     ===============
c     mat      (int,sc,in)         - material ID
c     plOpt    (int,sc,in)         - plasticity option
c     ncomp    (int,sc,in)         - number of strain components (1,4, or 6)

c     output arguments
c     ================
c     elLenPsvrBuf  (int,sc,out)   - number of extra data items saved

c     local variables
c     ===============

c**********************************************************************
```

2.2.21. Function tbgettbtype (Retrieving the Unique Index Associated with a TB Table)

```
*deck,tbgettbtype
      function tbgettbtype(tablename)
c**********************************************************************
c
c     *** primary function
c            Get the tbtype number associated with a string
c
c     input arguments
c     ===============
c     tabelename       char4,sc           unique table type string associated
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                     each type.
c     output arguments
c     ===============
c     tbgetdataperfield               tbtype unique to each type like 'user'
c                                     'elas', 'plas'
c
c*************************************************************************
```

2.2.22. Function tbgetnumtables (Retrieving the Number of Subtables for a TB Type)

```
*deck,tbgetnumtables
      function tbgetnumtables(matid,tabtyp)
c*************************************************************************
c
c     *** primary function
c            Get the number of user tb commands issued for mat id.
c
c     input arguments
c     ===============
c     matid        int,sc             material number
c     tabtyp       int,sc             unique table type number associated
c                                     each type. default is user when tabtyp
c                                     is set to a number less than 1
c
c     output arguments
c     ===============
c     tbgetnumtables                get number of tb table tables
c
c*************************************************************************
```

2.2.23. Function tbgetnumfldvars (Retrieving the Number of Field Variables for a TB Type)

```
*deck,tbgetnumfldvars
      function tbgetnumfldvars(matid,tabtyp,tbindx)
c*************************************************************************
c
c     *** primary function
c            Get number of field variables types
c
c     input arguments
c     ===============
c     matid        int,sc             material number
c     tabtyp       int,sc             unique table type number associated
c                                     each type. default is user when tabtyp
c                                     is set to a number less than 1
c     tbindx       int,sc             table index if multiple tb commands
c                                     for tb table were issued
c
c     output arguments
c     ===============
c     tbgetnumfldvars               get number of field variable types
c
c*************************************************************************
```

2.2.24. Function tbgetfldvars (Retrieving Field Variable Types for a TB Type)

```
*deck,tbgetfldvars
      function tbgetfldvars(matid,tabtyp,tbindx,fldtyp,nfld)
c*************************************************************************
c
c     *** primary function
c            Get field variables types indices. FLD_FieldDefines.inc has
c            the types and documentation. To see list of supported types
c            see the material reference manual
c
c     input arguments
c     ===============
c     matid        int,sc             material number
c     tabtyp       int,sc             unique table type number associated
c                                     each type. default is user when tabtyp
c                                     is set to a number less than 1
c     tbindx       int,sc             table index if multiple tb commands
c                                     for tb table were issued
c
c     output arguments
c     ================
c     tbgetnumfldvars               status 1 if success. 0 if failure
c     fldtyp                        Types of field variables.
c     nfld                          number of field variables.
c*************************************************************************
```

2.2.25. Function tbgetfldname (Retrieving a Field Variable Name)

```
*deck,tbgetfldname
      function tbgetfldname(fldtyp,fldstring)
c*************************************************************************
c
c     *** primary function
c            Get field variable name given an index or unique fld typ
c            queries using the tbgetfldvars function
c
c     input arguments
c     ===============
c     fldtyp       int,sc             field variable index
c
c     output arguments
c     ================
c     tbgetfldname               status 1 if success. 0 if failure
c     fldstring                  field variable name
c*************************************************************************
```

2.2.26. Function tbgettbopt (Retrieving the *TBOPT* Associated with a TB Type)

```
*deck,tbgettbopt
      function tbgettbopt(matid,tabtyp,tbindx)
c*************************************************************************
c
c     *** primary function
c            Get the tbopt of the subtable
c
c     input arguments
c     ===============
c     matid        int,sc             material number
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c     tabtyp       int,sc             unique table type number associated
c                                     each type. default is user when tabtyp
c                                     is set to a number less than 1
c     tbindx       int,sc             table index if multiple tb commands
c                                     for tb table were issued
c
c     output arguments
c     ===============
c     tbgettbopt                      tboption unique to each subtable
c
c***********************************************************************
```

2.2.27. Function tbgettboptname (Retrieving a String or Documented Name Associated with a *TBOPT*)

```
*deck,tbgettboptname
      function tbgettboptname(matid,tabtyp,tbindx,tboptstr)
c***********************************************************************
c
c     *** primary function
c            Get the string or documented name associated along with the
c            tbopt of a tb sub table
c
c     input arguments
c     ===============
c     matid        int,sc             material number
c     tabtyp       int,sc             unique table type number associated
c                                     each type. default is user when tabtyp
c                                     is set to a number less than 1
c     tbindx       int,sc             table index if multiple tb commands
c                                     for tb table were issued
c
c     output arguments
c     ===============
c     tbgettbopt                      tboption unique to each subtable
c     tboptstr                        string associated with the tbopt
c
c***********************************************************************
```

2.2.28. Function tbgetdataperfield (Retrieving the Data per Field for Each Subtable)

```
*deck,tbgetdataperfield
      function tbgetdataperfield(matid,tabtyp,tbindx)
c***********************************************************************
c
c     *** primary function
c            Get the number of table data issued per temperature
c
c     input arguments
c     ===============
c     matid        int,sc             material number
c     tabtyp       int,sc             unique table type number associated
c                                     each type. default is user when tabtyp
c                                     is set to a number less than 1
c     tbindx       int,sc             table index if multiple tb commands
c                                     for tb table were issued
c
c     output arguments
c     ===============
c     tbgetdataperfield               tboption unique to each subtable
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c
c*************************************************************************
```

2.2.29. Function tbgetnumfldsets (Retrieving the Number of TBFIELD+TBDATA/*TB0PT* Sets)

```
*deck,tbgetnumfldsets
c
      function tbgetnumfldsets(matid,tabtyp,tbindx)
c*************************************************************************
c
c     *** primary function
c            Get number of temperatures defined or field variables.
c
c     input arguments
c     ===============
c     matid        int,sc             material number
c     tabtyp       int,sc             unique table type number associated
c                                     each type. default is user when tabtyp
c                                     is set to a number less than 1
c     tbindx       int,sc             table index if multiple tb commands
c                                     for tb table were issued
c
c     output arguments
c     ===============
c     tbgetnumfldsets               get number of field variables/temps
c
c*************************************************************************
```

2.2.30. Function tbgettabledata (Retrieving All Table Data and Field Variable Values)

```
*deck,tbgettabledata
      function tbgettabledata(matid,tabtyp,tbindx,setid,
     x                        udata,ndata,ufld,nufld)
c*************************************************************************
c
c     *** primary function
c            Get the table data and temperature/fld vars
c
c     input arguments
c     ===============
c     matid        int,sc             material number
c     tabtyp       int,sc             unique table type number associated
c                                     each type. default is user when tabtyp
c                                     is set to a number less than 1
c     tbindx       int,sc             table index if multiple tb commands
c                                     for tb table were issued
c
c     output arguments
c     ===============
c     tbgetdataperfield               tboption unique to each subtable
c
c*************************************************************************
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.2.31. Function tbgettabledatasz (Retrieving All Table Data and Field Variable Sizes)

```
*deck,tbgettabledatasz
      function tbgettabledatasz(matid,tabtyp,tbindx,setid,
     x                          nfld,nrow,ncol)
c*********************************************************************
c
c     *** primary function
c            Get the table data and temperature/fld vars sizes
c
c     input arguments
c     ===============
c     matid        int,sc             material number
c     tabtyp       int,sc             unique table type number associated
c                                     each type. default is user when tabtyp
c                                     is set to a number less than 1
c     tbindx       int,sc             table index if multiple tb commands
c                                     for tb table were issued
c
c     output arguments
c     ===============
c     tbgettabledatasz              1 for success 0 for failure
c
c*********************************************************************
```

2.2.32. Function nlget (Retrieving Material Nonlinear Property Information)

```
*deck,nlget
      function nlget (mat,iprop,prop)
c *** primary function:    get a material non-linear property (TB) table.

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        variable (typ,siz,intent)    description
c        mat      (int,sc,in)       - material number
c        iprop    (int,sc,in)       - property number (tbpnum in tblecm)
c                                        use 13 for tb,user
c                                        use 14 for tb,nl

c     output arguments:
c        variable (typ,siz,intent)    description
c        nlget   (int,sc,out)       - number of property values
c        prop    (dp,ar(nlget),out) - vector of the property values
c                                      (the first 15(tbhdsz) items are a header,
c                                      given below.  The terms are defined in
c                                      tblecm.inc)
c        --- terms of the descriptor record:
c        header(1) = tbtyp
c        header(2) = tbtems
c        header(3) = temloc
c        header(4) = dprtem
c        header(5) = tbrow
c        header(6) = tbcol
c        header(7) = rowkey
c        header(8) = nxtloc
c        header(9) = nxttem
c        header(10) = temptr
c        header(11) = tbpt
c        header(12) = tbsiz
c        header(13) = tbopt
c        header(14) = hypopt
c        header(15) = tbnpts
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.2.33. Subroutine usereo (Storing Data in the nmisc Record)

```
*deck,usereo
      subroutine usereo (elem,iout,nbsvr,bsvr,nnrsvr,nrsvr,npsvr,psvr,
     x ncsvr,csvr,nusvr,usvr,nnode,nodes,xyz,vol,leng,time,
     x timinc,nutot,utot,maxdat,numdat,udbdat)
c
c *** primary function:  to call userou, which allows user to store
c                              data in nmisc record
c
c *** Notice - This file contains ANSYS Confidential information ***
c
c  input arguments:
c   variable (typ,siz,intent)       description
c    elem    (int,sc,in)           - element number
c    iout    (int,sc,in)           - output unit number
c    nbsvr   (int,sc,in)           - number of basic element variables
c    bsvr    (dp,ar(nbsvr),in)     - basic element variables
c    nnrsvr  (int,sc,in)           - number of nonlinear element variables
c    nrsvr   (dp,ar(nnrsvr),in)    - nonlinear element variables
c    npsvr   (int,sc,in)           - number of plasticity element variables
c    psvr    (dp,ar(npsvr),in)     - plasticity element variables
c    ncsvr   (int,sc,in)           - number of creep element variables
c    csvr    (dp,ar(ncsvr),in)     - creep element variables
c    nusvr   (int,sc,in)           - number of user-supplied element variables
c    usvr    (dp,ar(nusvr),in)     - user-supplied element variables
c    nnode   (int,sc,in)           - number of nodes
c    nodes   (int,ar(nnode),in)    - node numbers
c    xyz     (dp,ar(6,nnode),in)   - nodal coordinates and rotations (virgin)
c    vol     (dp,sc,in)            - element volume (or area if 2-d)
c    leng    (dp,sc,in)            - element length (beams,spars,etc)
c    time    (dp,sc,in)            - current time
c    timinc  (dp,sc,in)            - current sub step time increment
c    nutot   (int,sc,in)           - length of dof solution vector utot
c    utot    (dp,ar(nutot),in)     - solution vector
c    maxdat  (int,sc,in)           - size of user output array (3 x nnode)
c                                     actually, = ielc(nmnmup)
c                                     for contact element it is equale to nusvr
c                                     but it does not exceed 120
c
c  output arguments:
c   variable (typ,siz,intent)       description
c    numdat  (int,sc,out)          - number of user output items in array udbdat
c    udbdat  (dp,ar(maxdat),out)   - user output items to be placed at the end
c                                     of the nmisc record
c
```

2.2.34. Subroutine eldwrtL (Writing Element Data to a File)

```
*deck,eldwrtL
      subroutine eldwrtL (ielem,edtype,lcerstL,edindxL,nval,value)

c *** primary function:    output element data to result file

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem     (int,sc,in)        - element number
c        edtype    (int,sc,in)        - element data type (see elparm)
c        lcerstL   (LONG,sc,inout)    - pointer to results file position
c        edindxL   (LONG,ar(NUMELEDATASETS),inout)- index to results file data
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c        nval      (int,sc,in)        - the total number of values
c                                       if edtype = EDEMS,
c                                          this should -always- be ielc(nmsmis),
c                                          unless there is a variable number, as
c                                          in the layered shell elements.
c        value     (dp,ar(nval),in)   - output values (real)
```

2.2.35. Subroutine eldwrnL (Writing Element Nonsummable Miscellaneous Data to the Results File)

```
*deck,eldwrnL
      subroutine eldwrnL (elem,ielc,lcerstL,edindxL,nudb,udbdat,
     x                    nval,value,ndval)
c *** primary function:    output element nonsummable miscellaneous data
c                                    to result file
c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        elem      (int,sc,in)       - element number
c        ielc    (int,ar(IELCSZ),in) - element characteristic vector
c                                         defined in elccmt
c        lcerstL   (LONG,sc,inout)   - pointer to results file position
c        edindxL   (LONG,ar(NUMELEDATASETS),inout)- index to results file data
c        nudb      (in,sc,inout)     - size of what the user wants to add
c        udbdat    (dp,ar(*),in)     - what the user wants to add
c        nval      (int,sc,in)       - the total number of values to
c                                       be output(does not include nudb)
c                                       this should -always- be ielc(NMNMIS),
c                                       unless there is a variable number, as
c                                       in the layered shell elements.
c        value     (dp,ar(ndval),in) - output values
c        ndval     (int,sc,in)       - dimension of value - must be no less than
c                                        ielc(NMNMIS) + ielc(NMNMUP)
c *** mpg eldwrnL < el117,el126,el109,el53,el96,el97: write nmisc db
c
```

2.2.36. Subroutine trrot (Calculating the Rotation Vector)

```
*deck,trrot
      subroutine trrot (tr,rot)
c *** primary function:    get the rotation vector from a transformation matrix

c *** Notice - This file contains ANSYS Confidential information ***

c  input arguments:
c     tr       (dp,ar(3,3),in)     - transformation matrix

c  output arguments:
c     rot      (dp,ar(3),out)      - rotation vector
```

2.2.37. Subroutine rottr (Calculating the Transformation Matrix)

```
*deck,rottr
      subroutine rottr (rot,tr)
c primary function: compute transformation matrix from rotation vector *****
c *** Notice - This file contains ANSYS Confidential information ***
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c
c     ref(old): eqn. 20(transposed),rankin and brogan, jpvt,108(1986)165-174.
c     ref(new): eqn. (b.4), simo and vu-quoc, cmame, 58 (1986), 79-116
c                  (removes singularities at pi and 2*pi)
c
c  input arguments:
c     variable (typ,siz,intent)    description
c     rot      (dp,ar(4),in)     - rotation parameter in radians
c
c  output arguments:
c     variable (typ,siz,intent)    description
c     tr       (dp,ar(3,3),out)  - transformation matrix corresponding to rot
```

2.2.38. Subroutine xyzup3 (Updating an Element's 3-D Nodal Coordinates)

```
*deck,xyzup3
      subroutine xyzup3 (nnod,u,nr,xyz,nx,xyzup)
c *** primary function:  update a 3-d ele nodal coords for large deformation
c
c *** Notice - This file contains ANSYS Confidential information ***
c
c  input arguments:
c     nnod     (int,sc,in)          - number of nodes
c     u        (dp,ar(nr),in)       - displacement vector
c     nr       (int,sc,in)          - size of the u vector
c     xyz      (dp,ar(nx,nnod),in)  - coordinates to be updated
c     nx       (int,sc,in)          - row size of xy
c
c  output arguments:
c     xyzup     (dp,ar(3,nnod),out) - updated coordinates
c
```

2.2.39. Subroutine tmpget (Defining Current Temperature Loads)

```
*deck,tmpget
      subroutine tmpget (iel,ielc,nnod,nodes,ref,ndat0,begdat,dat,
     x   enddat,tlvf)
c     primary function: define the current temperature loads
c
c *** Notice - This file contains ANSYS Confidential information ***
c
c  typ=int,dp,log,chr,dcp   siz=sc,ar(n),func    intent=in,out,inout
c
c  input arguments:
c  variable (typ,siz,intent)     description
c     iel    (int,sc,in)         - element number
c     ielc   (int,ar(IELCSZ),in) - array of element type characteristics
c     nnod   (int,sc,in)         - number of nodes in the nodes array
c     nodes  (int,ar(nnod),in)   - list of nodes
c     ref    (dp,sc,in)          - reference temperature
c     ndat   (int,sc,in)         - number of data items to get
c     begdat (dp,ar(ndat),in)    - data at the beginning of this load step
c
c  output arguments:
c     dat    (dp,ar(ndat),out)   - data at this time point
c     enddat (dp,ar(ndat),out)   - data at end of this load step
c     tlvf   (int,sc,out)        - thermal load vector flag
c                                  Should the thermal load vector be computed
c                                  = 0 - no, temperatures match tref
c                                  = 1 - yes, temperatures do not match tref
c                                  =<0 - no and using table
c                   Note, that even if tlvf = 0, temperatures may be used to
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                   compute temperature-dependent material properties.
c
```

2.2.40. Subroutine prsget (Defining Current Pressure Loads)

```
*deck,prsget
      subroutine prsget (iel,ielc,nfac,ndat,begdat,dat,enddat,iexist)

c     primary function: define the current pressure loads

c     See also:  PrsRIGet

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        iel      (int,sc,in)      - element number
c        ielc  (int,ar(IELCSZ),in) - array of element type characteristics
c        nfac     (int,sc,in)      - number of pressure faces
c        ndat     (int,sc,in)      - number of pressure values
c        begdat   (dp,ar(ndat),in) - pressure at the beginning of load step

c     output arguments:
c        dat      (dp,ar(ndat),out) - pressures at this iteration
c        enddat   (dp,ar(ndat),out) - pressure at end of this load step
c        iexist   (int,sc,out)      - flag if pressure exist
c                                     =  0 : no pressure
c                                     =  1 : yes pressure
c                                     = -1 : no pressure and has table
```

2.2.41. Subroutine cnvget (Defining Current Convection Loads)

```
*deck,cnvget
      subroutine cnvget (iel,ielc,nr,u,nfac,ndat,beghc,begtb,
     x   hc,tb,endhc,endtb,iexist)
c     primary function: define the current convection loads

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        iel      (int,sc,in)      - element number
c        ielc  (int,ar(IELCSZ),in) - array of element type characteristics
c        nr       (int,sc,in)      - dimension of u (temperature) vector
c        u        (dp,ar(nr),in)   - most current temperatures
c        nfac     (int,sc,in)      - number of convection faces
c        ndat     (int,sc,in)      - number of convection values
c        beghc    (dp,ar(ndat),in) - hcoef at the beginning of load step
c        begtb    (dp,ar(ndat),in) - tbulk at the beginning of load step

c     output arguments:
c        hc       (dp,ar(ndat),out) - hcoef at this substep
c        tb       (dp,ar(ndat),out) - tbulk at this substep
c        endhc    (dp,ar(ndat),in)  - hcoef at the end of this load step
c        endtb    (dp,ar(ndat),in)  - tbulk at the end of this load step
c        iexist   (int,sc,out)      - flag if convection exist
c                                     = 0 - no convection
c                                     = 1 - constant convection (with time)
c                                           does not require new element matrix
c                                     = 2 - changing convection (with time)
c                                           or deleted convection
c                                           requires new element matrix
```

2.2.42. Subroutine hgnget (Defining Current Heat Generation Loads)

```
*deck,hgnget
      subroutine hgnget (iel,ielc,nnod,nodes,ndat,begdat,dat,enddat,
     x   iexist)
c     primary function: define the current heat generation loads
c
c *** Notice - This file contains ANSYS Confidential information ***
c
c     typ=int,dp,log,chr,dcp   siz=sc,ar(n),func    intent=in,out,inout
c
c     input arguments:
c     variable (typ,siz,intent)     description
c        iel    (int,sc,in)         - element number
c        ielc   (int,ar(IELCSZ),in) - array of element type characteristics
c        nnod   (int,sc,in)         - number of nodes in the nodes array
c        nodes  (int,ar(nnod),in)   - list of nodes
c        ndat   (int,sc,in)         - number of data items to get
c        begdat (dp,ar(ndat),in)    - data at the beginning of this load step
c
c     output arguments:
c        dat    (dp,ar(ndat),out)   - data at this time point
c        enddat (dp,ar(ndat),out)   - data at end of this load step
c        iexist (int,sc,out)        - flag if heat generation exist
c                                     = 0 - no heat generation
c                                     = 1 - yes heat generation
c
```

2.2.43. Subroutine prinst (Calculating Principal Stress and Stress Intensity)

```
*deck,prinst
      subroutine prinst (s)
c    primary function:  computes principal stresses and stress intensity
c    secondary functions:  none
c *** Notice - This file contains ANSYS Confidential information ***

c  input arguments:
c     variable (typ,siz,intent)    description
c     s        (dp,ar(11),inout)   - stress vector
c                 s(1)=sx
c                 s(2)=sy
c                 s(3)=sz
c                 s(4)=sigxy
c                 s(5)=sigyz
c                 s(6)=sigzx

c
c  output arguments:
c     variable (typ,siz,intent)    description
c     s        (dp,ar(11),inout)   - stress vector
c                 s(7)=sig1
c                 s(8)=sig2
c                 s(9)=sig3
c                 s(10)=s.i.
c                 s(11)=sige

c ********* note: all changes to this routine must be made in
c                post1  (paprst)
c
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.3. Subroutines for Modifying and Monitoring Existing Elements

Following are the user subroutines for modifying or monitoring existing elements:

2.3.1. Subroutine userou (Storing User-Provided Element Output)

```
*deck,userou                       USERDISTRIB
      subroutine userou (elem,iout,nbsvr,bsvr,nnrsvr,nrsvr,npsvr,psvr,
     x ncsvr,csvr,nusvr,usvr,nnode,nodes,xyz,vol,leng,time,
     x timinc,nutot,utot,maxdat,numdat,udbdat)
c
c *** primary function:    store user supplied element output
c                           in nmisc record
c
c         in order to activate this user programmable feature,
c         the user must enter the usrcal command.
c
c
c         *** Copyright ANSYS.  All Rights Reserved.
c         *** ansys, inc.
c *** Notice - This file contains ANSYS Confidential information ***
c
c             this routine is called by almost every element
c             the data is stored on the nmisc record.
c             warning:  other data may be stored between the
c                       documented data and this data.
c             in order to see the actual information on the nmisc
c              record, insert the command:
c                  dblist,elp,elnum1,elnum2,elinc,11
c                       where elnum1 = the first element
c                             elnum2 = the last element
c                             elinc  = the element increment number
c              after a set command in post1.
c
c  input arguments:
c   variable (typ,siz,intent)       description
c    elem    (int,sc,in)           - element number
c    iout    (int,sc,in)           - output unit number
c    nbsvr   (int,sc,in)           - number of basic element variables
c    bsvr    (dp,ar(nbsvr),in)     - basic element variables
c    nnrsvr  (int,sc,in)           - number of nonlinear element variables
c    nrsvr   (dp,ar(nnrsvr),in)    - nonlinear element variables
c    npsvr   (int,sc,in)           - number of plasticity element variables
c    psvr    (dp,ar(npsvr),in)     - plasticity element variables
c    ncsvr   (int,sc,in)           - number of creep element variables
c    csvr    (dp,ar(ncsvr),in)     - creep element variables
c    nusvr   (int,sc,in)           - number of user-supplied element variables
c                                         (= nstv on the nsvr command)
c    usvr    (dp,ar(nusvr),in)     - user-supplied element variables
c    nnode   (int,sc,in)           - number of nodes
```

```
c    nodes   (int,ar(nnode),in)    - node numbers
c    xyz     (dp,ar(6,nnode),in)   - nodal coordinates and rotations (virgin)
c    vol     (dp,sc,in)            - element volume (or area if 2-d)
c    leng    (dp,sc,in)            - element length (beams,spars,etc)
c    time    (dp,sc,in)            - current time
c    timinc  (dp,sc,in)            - current sub step time increment
c    nutot   (int,sc,in)           - length of dof solution vector utot
c    utot    (dp,ar(nutot),in)     - solution vector
c    maxdat  (int,sc,in)           - size of user output array (3 x nnode)
c                                     for contact element it is equale to nusvr
c                                     but it dode not exceed 120
c
c  output arguments:
c   variable (typ,siz,intent)       description
c    numdat  (int,sc,out)          - number of user output items in array udbdat
c                                      (maximum size of numdat is ielc(NMNMUP)
c                                      which is usually three times the number
c                                      of nodes.
c                                      For contact elements CONTA171-178,it
c                                      should be equal or less than NSTV
c                                      on nsvr command). It cannot exceed 120.
c    udbdat  (dp,ar(maxdat),out)   - user output items to be placed at the end
c                                    of the nmisc record
c
```

2.3.2. Subroutine useran (Modifying Orientation of Material Properties)

```
*deck,useran                       USERDISTRIB
      subroutine useran  (vn,vref,elem,thick,xyzctr,bsangl)
c     user written routine to modify orientation of material properties
c     and stresses **************************
c       applicable to: shell43,63,91,93,99, solid46,64,191
c        accessed by keyopt
c
c         *** Copyright ANSYS.  All Rights Reserved.
c         *** ansys, inc.
c *** Notice - This file contains ANSYS Confidential information ***
c
c **** warning *** do not change any arguments other than bsangl.
c                  if you do, your results are probably wrong.
c
c  input(do not change)---
c     vn       = vector normal to element
c     vref     = unit vector orienting element, essentially edge i-j
c     elem     = element number
c     thick    = total thickness of element at this point (see note below)
c     xyzctr = location of element centroid or integration point
c
c  output---
c     bsangl = output from this subroutine.  it represents the angle(s)
c              between vref and the desired orientation.  it may have
c              the default orientation coming in to useran.
c                This will be combined with the angles derived from
c                the ESYS command.
c           use 1 angle for 2-d elements and shells
c           use 3 angles for 3-d solids
c
```

2.3.3. Subroutine userrc (Performing User Operations on COMBIN37 Parameters)

```
*deck,userrc                       USERDISTRIB
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
      subroutine userrc (elem,ireal,type,nusvr,usvr,parm,parmld,
     x c1,c2,c3,c4,fcon)
c     primary function: user operation on parameter for combin37
c      accessed with keyopt(9) = 1
c
c         *** Copyright ANSYS.  All Rights Reserved.
c         *** ansys, inc.
c *** Notice - This file contains ANSYS Confidential information ***
c
c  input arguments:
c   variable (typ,siz,intent)       description
c    elem    (int,sc,in)           - element number
c    ireal    (int,sc,in)          - element real constant number
c    type    (int,sc,in)           - element type number
c    nusvr   (int,sc,in)           - number of user-supplied element variables
c                                     (input with the NSVR command)
c    usvr    (dp,ar(nusvr),inout)  - user-supplied element variables
c    parm    (dp,sc,in)            - current value of the paramater
c    parmld  (dp,sc,in)            - value of the parameter at previous time ste
c    c1      (dp,sc,in)            - real constant c1
c    c2      (dp,sc,in)            - real constant c2
c    c3      (dp,sc,in)            - real constant c3
c    c4      (dp,sc,in)            - real constant c4
c
c  output arguments:
c   variable (typ,siz,intent)       description
c    usvr    (dp,ar(nusvr),inout)  - user-supplied element variables
c                                     may be sent .rst file with usereo
c    fcon    (dp,sc,out)           - result of calculation
c
c     either c1 or c3 must be nonzero for this logic to be accessed,
c
```

2.3.4. Subroutine UElMatx (Accessing Element Matrices and Load Vectors)

```
*deck,UElMatx                       USERDISTRIB
      subroutine UElMatx (elem,nr,ls,zs,zsc,uelm,ielc,nodes,
     x                    ElDofEachNode,elmdat,xyzang,lenu)

c primary function:    User routine to access element matrices and load vectors.
c                      Needs to have USRCAL,UELMATX to be accessed.
c                      Called after the call to the element routine and
c                       before the solver.
c                      May be used to monitor and/or modify the element matrices
c                                                 and load vectors.

c         *** Copyright ANSYS.  All Rights Reserved.
c         *** ansys, inc.

c     typ=int,dp,log,chr,dcp   siz=sc,ar(n)   intent=in,out,inout

c  input arguments:
c     variable (typ,siz,intent)        description
c     elem     (int,sc,in)           - User element number
c     nr       (int,sc,in)           - number of rows in element matrix
c     ls       (int,ar(nr),in)       - Dof Index vector for this element matrix
c     zs       (dp,ar(nr,nr,*),inout)- K,M,C,SS,KCPLX matrices for this element
c     zsc      (dp,ar(nr,2),inout)   - Element load vector and N-R correction vec
c     uelm     (dp,ar(nr,5),in)      - Nodal displacements for this element
c     ielc     (int,ar(*),in)        - Element type characteristics
c     nodes    (int,ar(*),in)        - Nodes for this element
c     ElDofEachNode (int,ar(nr),in) - list of dofs for each node in Global
c     elmdat   (int,ar(10),in)       - Element data for this element
c     xyzang   (dp,ar(6,*),in)       - X,Y,Z,THXY,THYZ,THZX for each element node
c     lenu     (int,sc,in)           - Length of global displacement vector

c  output arguments:
```

```
c     zs       (dp,ar(nr,nr,4),inout)- K,M,C,SS matrices for this element
c     zsc      (dp,ar(nr,2),inout)  - Element load vector and N-R correction vec
c      WARNING:  any CHANGES to these (or any other) arguments will have a direc
c      impact on the solution, possibly giving meaningless results.  The normal
c      usage of this routine is simply monitor what is happening.
```

2.3.5. Subroutine uthick (Getting User-Defined Initial Thickness)

```
*deck,uthick                      USERDISTRIB
      SUBROUTINE uthick (elemId, elemType, matId, realId,
     $                   numDomIntPts, curCoords, thickness)
ccccccccccccccccccccccccccccccccccccccccccccccccccccccccccccccccccccccccc
c
c     *** primary function: get the user defined thickness
c
c         *** Copyright ANSYS.  All Rights Reserved.
c         *** ansys, inc.
c
c     input arguments
c     ===============
c     Variable        (type,sz,i/o)  description
c      elemId         (int,sc,i)     element number
c      elemType       (int,sc,i)     element TYPE (181 etc.)
c      matId          (int,sc,i)     material number
c      realId         (int,sc,i)     real constant set number
c      numDomIntPts   (int,sc,i)     number of integration points
c      curCoords      (dp,ar(3,numDomIntPts),i)
c                                    current coordinates
c
c     output arguments
c     ================
c      thickness      (dp,ar(3,numDomIntPts),o)
c                                    thickness at the integration points
c
ccccccccccccccccccccccccccccccccccccccccccccccccccccccccccccccccccccccccc
c
c --- parameters
c
```

2.3.6. Subroutine uflex (Calculating Flexibility Factors for PIPE288 and PIPE289)

```
*deck,uflex                       USERDISTRIB
      subroutine uflex (elemId,pressInt,pressExt,ex,pois,  sflex,twten)
c *** primary function:   to (re)compute the flexibility factors
c                           for pipe288 and pipe289
c                         this is accessed by inputting the axial flexibility factor
c                           as -10.
c *** secondary functions: none
c
c *** Notice - This file contains ANSYS Confidential information ***
c
c         *** Copyright ANSYS.  All Rights Reserved.
c         *** ansys, inc.
c
c     typ=int,dp,log,chr,dcp   siz=sc,ar(n)   intent=in,out,inout
c
c     input arguments:
c  elemId      (int,sc,in) - element number
c  pressInt  (dp,ar(2),in) - internal pressures at end nodes
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c  pressExt  (dp,ar(2),in) - external pressures at end nodes
c                                    Pressures include hydrostatatic but
c                                     not hydrodynamic effects.
c  ex          (dp,sc,in) - Young's Modulus
c  pois        (dp,sc,in) - Poisson's ratio
c  sflex  (dp,ar(6),inout) - input flexibility factors
c                             (axial, bending about element z,
c                              bending about element y, twist, y shear, z shear)
c  twten     (dp,sc,inout) - twist-tension factor
c
c     output arguments:
c  sflex  (dp,ar(6),inout) - output flexibility factors
c                             (axial, bending about element z,
c                              bending about element y, twist, y shear, z shear)
c  twten     (dp,sc,inout) - twist-tension factor
c
```

2.3.7. Subroutine UsrShift (Calculating Pseudotime Time Increment)

```
*deck,UsrShift                       USERDISTRIB
c Copyright ANSYS.  All Rights Reserved.
      subroutine UsrShift(dxi,dxihalf,timinc,
     &                    temp,dtemp,toffst,propsh,nTerms)
c*************************************************************************
c     calculate pseudotime time increment according
c     to a user specified shift function
c
c *** Notice - This file contains ANSYS Confidential information ***
c
c  input arguments:
c     timinc  (dp,sc,in)         - time increment
c     temp    (dp,sc,in)         - current temperature, t_n+1
c     dtemp   (dp,sc,in)         - temperature increment, t_n+1 - t_n
c     toffst  (dp,sc,in)         - temperature offset to absolute zero
c                                  (specified by TOFFST command)
c     propsh  (dp,ar,in)         - Constants for shift function
c                                  (User's input using TB,SHIFT,,,,USER)
c     nTerms  (int,ar,in)        - number of user defined constants
c                                  (specified in TB,SHIFT,,,nTerms,USER)
c  output arguments:
c    dxi      (dp,sc,out)        - pseudotime increment
c    dxihalf  (dp,sc,out)        - pseudotime increment over the upper half span
c*************************************************************************
```

2.3.8. Subroutine UTimeInc (Overriding the Program-Determined Time Step)

This subroutine allows you to create a user-defined time step to override the one determined by the program. Activate the subroutine via the **USRCAL**,UTIMEINC command.

```
*deck,UTimeInc                       USERDISTRIB
      subroutine UTimeInc (deltmin,deltmax,delt)

c primary function:    User routine to override the program determined time step
c                      Needs to have USRCAL,UTIMEINC to be accessed.
c                      Called after the program determined the next time step
c                        increment (AUTOTS,ON only)

c         *** Copyright ANSYS.  All Rights Reserved.
c         *** ansys, inc.

c  input arguments:
c     deltmin  (int,dp,in)    - minimum time step size (user input)
```

```
c     deltmax  (int,dp,in)    - maximum time step size (user input)
c     delt     (int,dp,inout) - on input, the value determined by the program

c  output arguments:
c     delt     (int,dp,inout) - on output, the value you have determined
```

2.3.9. Subroutine UCnvrg (Overriding the Program-Determined Convergence)

This subroutine allows you to create user-defined convergence checking and to override the convergence determined by the program. Activate the subroutine via the **USRCAL**,UCNVRG command.

```
*deck,UCnvrg                       USERDISTRIB
      subroutine UCnvrg (ConvergenceType,ConvergenceFlag)

c primary function:    User routine to perform custom convergence checking and
c                        override the program-determined convergence
c                      Needs to have USRCAL,UCNVRG to be accessed.
c                      Called after the program convergence checks.

c         *** Copyright ANSYS.  All Rights Reserved.
c         *** ansys, inc.

c  input arguments:
c     ConvergenceType  (int,sc,in)    - type of convergence to be checked
c                                         1, nonlinear element (called after
c                                            element matrix formation)
c                                         2, force convergence (called after
c                                            element matrix formation)
c                                         3, displacement convergence (called after
c                                            equation solution)
c     ConvergenceFlag  (int,sc,inout) - on input, the value the program determined
c                                       for this Type
c                                         0, not converged
c                                         1, converged

c  output arguments:
c     ConvergenceFlag  (int,sc,inout) - on output, the value the you have determined
c                                       for this Type
c                                         0, not converged
c                                         1, converged

c     Note: For overall convergence, all 3 Types must be converged. Not all
c           Types are evaluated (dependent on CNVTOL input and program defaults)
```

2.4. Subroutines for Customizing Material Behavior

This section describes the following subroutines that you can use to modify or monitor material behavior:

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Using the "_MATL" String

If you write a material-behavior subroutine using the **MPDATA**, **MPDELE**, **TB**, or **TBDELE** command, be aware that when the string "_MATL" appears in the *MAT* field of the command, the command interprets the string to mean the currently active material (as defined via the **MAT**,*MAT* command).

The "_MATL" string is used in conjunction with the library (LIB) option of the **MPREAD** and **MPWRITE** commands. When you issue **MPWRITE** with the LIB option, the command inserts "_MATL" in lieu of the specified material number as these commands are written to the material library file. When the program reads a material library file written in this format, it interprets "_MATL" to mean the currently active material. Do not use the "_MATL" string outside the scope of the **MPREAD** command.

2.4.1. Subroutine UserMat (Creating Your Own Material Model)

The `UserMat` subroutine allows you to write your own material constitutive equations within a general material framework using current-technology elements.

`UserMat` is a tool for advanced users. Expertise in material constitutive modeling and software programming is necessary. Developing a custom material constitutive model requires validation and testing. Ansys, Inc. strongly recommends testing both single elements and multiple elements with various loading conditions to ensure correct results. `UserMat` supports shared memory and distributed parallel processing; however, you are responsible for ensuring that your code can use parallel processing.

The following `UserMat` topics are available:

For a `UserMat` subroutine example, see Appendix C: User Material (UserMat) Subroutine Example (p. 395). For an example of UserMat in a coupled-field analysis, see Appendix D: Structural-Thermal User Material (UserMat, UserMatTh) Example (p. 405).

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.4.1.1. UserMat Element Support

Element support for user-defined material models (**TB**,USER) is available in Material Model Element Support in the *Material Reference*.

2.4.1.2. UserMat Overview

The `UserMat` subroutine defines the material stress-strain relationship of a material and applies to time-domain and full-harmonic analysis types. The subroutine supports current-technology elements only and does not apply to legacy elements.

The subroutine is called at every material integration point of the elements during the solution phase. The program passes in stresses, strains, and state variable values at the beginning of the increment (and for time-domain analyses, the strain increment at the current increment). The `UserMat` subroutine then updates the stresses and state variables to current values.

2.4.1.2.1. Time Domain

The two most important `UserMat`-calculated quantities are stress and the consistent tangent stiffness:

- The measure of the stress is Cauchy stress (true stress), σ.
- The consistent tangent stiffness is defined as $\partial \Delta \boldsymbol{\sigma} / \partial \Delta \boldsymbol{\varepsilon}$, where $\Delta \boldsymbol{\sigma}$ is the stress increment and $\Delta \boldsymbol{\varepsilon}$ is the strain increment. The measure of strain is logarithmic strain (true strain), $\boldsymbol{\varepsilon}$.

Generally, the material model can be formulated in either incremental (rate) or total forms.

- The incremental form is suitable for material models such as plasticity. The program uses a co-rotated framework for such material formulation. All variables are defined and updated in the co-rotated material coordinate system. The implementation of a constitutive model looks exactly the same for both small and large deformation (**NLGEOM**,ON).
- The total form is commonly used with hyperelastic material, and the material response is characterized by a potential relating to the invariants of the deformation gradient. The discrete equations are formulated with respect to the reference configuration. The deformation gradient is defined in the global coordinate system (with the exception of shell elements, where the deformation gradient is expressed on the co-rotated element coordinate system).

For most of the hyper-elastic models, material behavior is considered to be incompressible or nearly incompressible, resulting in a singular element matrix and possibly leading to element volumetric locking and solution failure. It is therefore necessary to take measures to avoid the singularity. The penalty method of carefully selecting the penalty parameter (which is also the compressibility parameter) is the simplest approach. Elements with mixed u-P formulation are the best way to address the singularity caused by material incompressibility. To use this option, set *TB0PT* = MXUP and element KEYOPT(6) = 1.

For a user-defined material with purely incompressible behavior, a mixed formulation in which the pressure is a degree of freedom is necessary. When a nearly incompressible hyperelastic material is formulated in `UserMat`, additional information (derivatives of volumetric potential with regard to determinant of deformation gradient J) is required. For more information, see UserMat Variables (p. 211).

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Input values and the number of state variables (if used) for UserMat are specified via the **TB** command. For more information, see Table (**TB**) Commands for UserMat (p. 214).

2.4.1.2.2. Harmonic

For full-harmonic analyses, the incoming strain argument is a two-dimensional vector containing the real (material stiffness) and imaginary (damping) parts of the harmonic strain. The subroutine must calculate and return the real and imaginary parts of the stress and material Jacobian matrix for the current frequency.

The complex stress returned by the subroutine has no effect on the harmonic solution and is used for postprocessing only.

2.4.1.3. Stress, Strain, and Material Jacobian Matrix

For nonlinear geometry analysis, the stress measure (σ) used by the subroutine is the Cauchy stress (true stress), and the strain measure (ε) is the logarithmic strain (true strain). The strains and incremental strains passed into UserMat are the total mechanical strains from which the thermal strains (if they exist) are subtracted.

UserMat must also provide the material Jacobian matrix defined as $\partial \Delta\sigma_{ij} / \partial \Delta\varepsilon_{ij}$. $\Delta\sigma_{ij}$ is the stress increment, and $\Delta\varepsilon_{ij}$ is the strain increment.

UserMat is based on the current configuration for nonlinear geometry analysis (**NLGEOM**,ON). The program uses a co-rotational approach to account for rigid body rotation. Because the program already accounts for the strains passed into UserMat for the rigid body rotation, there is no need to apply additional rotation within UserMat.

Stress, strain, and the material Jacobian tensors are stored in a vector or matrix format.

The order of components for all tensors is as follows:

3-D stress state

11, 22, 33, 12, 23, 13

2-D plane strain and axisymmetric stress states

11, 22, 33, 12

2-D plane stress states

11, 22, 12

Beam element stress states

11, 13, 12

Link element stress state

11

The order of components for the material Jacobian matrix is as follows:

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3-D stress state

1111	1122	1133	1112	1123	1113
2211	2222	2233	2212	2223	2213
3311	3322	3333	3312	3323	3313
1211	1222	1233	1212	1223	1213
2311	2322	2333	2312	2323	2313
1311	1322	1333	1312	1323	1313

2-D plane strain and axisymmetric stress states

1111	1122	1133	1112
2211	2222	2233	2212
3311	3322	3333	3312
1211	1222	1233	1212

2-D plane stress states

1111	1122	1112
2211	2222	2212
1211	1222	1212

Beam element stress states

1111	1113	1112
1311	1313	1312
1211	1213	1212

Link element stress state

1111

2.4.1.4. The UserMat API

Following is the interface for the `UserMat` subroutine:

```
*deck,usermat                       USERDISTRIB
      subroutine usermat(
     &                   matId, elemId,kDomIntPt, kLayer, kSectPt,
     &                   ldstep,isubst,keycut,
     &                   nDirect,nShear,ncomp,nStatev,nProp,
     &                   Time,dTime,Temp,dTemp,
     &                   stress,ustatev,dsdePl,sedEl,sedPl,epseq,
     &                   Strain,dStrain, epsPl, prop, coords,
     &                   var0, defGrad_t, defGrad,
     &                   tsstif, epsZZ,
     &                   cutFactor, pVolDer, hrmflg, var3, var4,
     &                   var5, var6, var7)
c*************************************************************************
c     *** primary function ***
c
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c           user defined material constitutive model
c
c      Attention:
c           User must define material constitutive law properly
c           according to the stress state such as 3D, plane strain
c           and axisymmetry, plane stress and 3D/1D beam.
c
c           A 3D material constitutive model can be used for
c           plane strain and axisymmetry cases.
c
c           When using shell elements, a plane stress algorithm
c           must be used.
c
c                                          gal July, 1999
c
c       The following demonstrates a USERMAT subroutine for
c       a plasticity model, which is the same as TB, BISO,
c       for different stress states.
c       See "ANSYS user material subroutine USERMAT" for detailed
c       description of how to write a USERMAT routine.
c
c       This routine calls four routines,
c       usermat3d.F, usermatps.F usermatbm.F and usermat1d.F, w.r.t.
c       the corresponding stress states.
c       Each routine can be also a usermat routine for the specific
c       element.
c
c*************************************************************************
c Copyright ANSYS.  All Rights Reserved.
c
c     input arguments
c     ===============
c      matId     (int,sc,i)              material #
c      elemId    (int,sc,i)              element #
c      kDomIntPt (int,sc,i)              "k"th domain integration point
c      kLayer    (int,sc,i)              "k"th layer
c      kSectPt   (int,sc,i)              "k"th Section point
c      ldstep    (int,sc,i)              load step number
c      isubst    (int,sc,i)              substep number
c      nDirect   (int,sc,in)             # of direct components
c      nShear    (int,sc,in)             # of shear components
c      ncomp     (int,sc,in)             nDirect + nShear
c      nstatev   (int,sc,i)              Number of state variables
c      nProp     (int,sc,i)              Number of material constants
c
c      Temp      (dp,sc,in)              temperature at beginning of
c                                        time increment
c      dTemp     (dp,sc,in)              temperature increment
c      Time      (dp,sc,in)              time at beginning of increment (t)
c      dTime     (dp,sc,in)              current time increment (dt)
c
c      Strain   (dp,ar(ncomp),i)         Strain at beginning of time increment
c      dStrain  (dp,ar(ncomp),i)         Strain increment
c      prop     (dp,ar(nprop),i)         Material constants defined by TB,USER
c      coords   (dp,ar(3),i)             current coordinates
c      defGrad_t(dp,ar(3,3),i)           Deformation gradient at time t
c      defGrad  (dp,ar(3,3),i)           Deformation gradient at time t+dt
c      hrmflg   (dp,sc,io)               flag to indicate harmonic analysis
c
c     input output arguments
c     ======================
c      stress   (dp,ar(ncomp),io)        stress
c      ustatev   (dp,ar(nstatev),io)     user state variables
c      sedEl    (dp,sc,io)               elastic work
c      sedPl    (dp,sc,io)               plastic work
c      epseq    (dp,sc,io)               equivalent plastic strain
c      epsPl   (dp,ar(ncomp),io)         plastic strain
c      var?     (dp,sc,io)               not used, they are reserved arguments
c                                        for further development
c
c     output arguments
```

```
c     ================
c      keycut   (int,sc,o)               loading bisect/cut control
c                                        0 - no bisect/cut
c                                        1 - bisect/cut
c                                        (factor will be determined by solution control)
c      dsdePl   (dp,ar(ncomp,ncomp),o)   material jacobian matrix
c      pVolDer  (dp,ar(3),o)             derivatives of volumetric potential wrt to J
c                                        pVolDer(1) = dU/dJ
c                                        pVolDer(2) = d^2U/dJ^2
c                                        pVolDer(3) = d^3U/dJ^3
c      tsstif   (dp,ar(2),o)             transverse shear stiffness
c                                        tsstif(1) - Gxz
c                                        tsstif(2) - Gyz
c                                        tsstif(1) is also used to calculate hourglass
c                                        stiffness, this value must be defined when low
c                                        order element, such as 181, 182, 185 with uniform
c                                        integration is used.
c      epsZZ    (dp,sc,o)                strain epsZZ for plane stress,
c                                        define it when accounting for thickness change
c                                        in shell and plane stress states
c      cutFactor(dp,sc,o)                time step size cut-back factor
c                                        define it if a smaller step size is wished
c                                        recommended value is 0~1
c
c*********************************************************************
c
c      ncomp   6   for 3D  (nshear=3)
c      ncomp   4   for plane strain or axisymmetric (nShear = 1)
c      ncomp   3   for plane stress (nShear = 1)
c      ncomp   3   for 3d beam      (nShear = 2)
c      ncomp   1   for 1D (nShear = 0)
c
c      stresses and strains, plastic strain vectors
c          11, 22, 33, 12, 23, 13    for 3D
c          11, 22, 33, 12            for plane strain or axisymmetry
c          11, 22, 12                for plane stress
c          11, 13, 12                for 3d beam
c          11                        for 1D
c
c      material jacobian matrix
c        3D
c           dsdePl    |  1111   1122   1133   1112   1123   1113 |
c           dsdePl    |  2211   2222   2233   2212   2223   2213 |
c           dsdePl    |  3311   3322   3333   3312   3323   3313 |
c           dsdePl    |  1211   1222   1233   1212   1223   1213 |
c           dsdePl    |  2311   2322   2333   2312   2323   2313 |
c           dsdePl    |  1311   1322   1333   1312   1323   1313 |
c        plane strain or axisymmetric (11, 22, 33, 12)
c           dsdePl    |  1111   1122   1133   1112 |
c           dsdePl    |  2211   2222   2233   2212 |
c           dsdePl    |  3311   3322   3333   3312 |
c           dsdePl    |  1211   1222   1233   1212 |
c        plane stress (11, 22, 12)
c           dsdePl    |  1111   1122   1112 |
c           dsdePl    |  2211   2222   2212 |
c           dsdePl    |  1211   1222   1212 |
c        3d beam (11, 13, 12)
c           dsdePl    |  1111   1113   1112 |
c           dsdePl    |  1311   1313   1312 |
c           dsdePl    |  1211   1213   1212 |
c        1d
c           dsdePl    |  1111 |
c
c*********************************************************************
```

The ncomp value used in `get_ElmData` (p. 216) and `put_ElmData` (p. 403) may differ from the ncomp value passed into `usermat.F`. Use `get_ElmData` with the 'NCOMP' option to determine the correct array sizes.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.4.1.5. UserMat Variables

The `UserMat` subroutine uses the following Input (p. 211), Input/Output (p. 212), and Output (p. 213) variables. Do not change them in the subroutine code.

`UserMat` Input Arguments	
matId	Integer variable containing the material ID number.
elemId	Integer variable containing the element number.
kDomIntPt	Integer variable containing the material integration point number.
kLayer	Integer variable containing the layer number.
kSectPt	Integer variable containing section point number.
ldstep	Integer variable containing load step number.
isubst	Integer variable containing substep number.
nDirect	Number of direct components of the stress or strain vector at material point.
nShear	Number of shear components of the stress or strain vector at material point (engineering components).
ncomp	Total number of the stress or strain components at material point (nDirect + nShear).
nstatev	Number of state variables, specified by the *NPTS* value in the **TB**,STATE command.
nProp	Number of material constants, specified by the *NPTS* value in the **TB**,USER command.
Temp	Double-precision variable containing the temperature at the beginning of time increment.
dTemp	Double-precision variable containing the current temperature increment.
Time	For time-domain analyses, a double-precision variable containing the total time at the beginning of the time increment. For full-harmonic analyses, the current value of frequency.
dTime	For time-domain analyses, a double-precision variable containing the current time increment. For full-harmonic analyses, the frequency increment.
Strain	Double-precision array containing the total strains at the beginning of the time increment. Array size is Strain(ncomp) in time-domain analyses and Strain(ncomp,2) in harmonic analyses, where the real and imaginary strain components are in the first and second column, respectively. Thermal strains (defined via **MP**,ALPHA and temperature load), if any, are subtracted from the total strains; therefore, the strains passed to `UserMat` are the mechanical strains only. For large-deformation problems, (**NLGEOM**,ON), the strain components are updated to account for rigid body rotation before they are passed to `UserMat` and are approximately the logarithmic strains. When the mixed u-P formulation option (*TBOPT* = MXUP on the **TB**,USER command) is used for hyperelastic material, the strain array is the logarithmic strains at the current time. However, the strain array can be redefined within the `UserMat` subroutine. For nearly incompressible hyperelastic material, a mixed u-J formulation is used. The calculated J is passed from strain array as strain(ncomp+1).

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

dStrain	Double-precision array containing current strain increments. Array size is ncomp. As with the Strain array, this value contains the mechanical strain increments only. Thermal strain increments (if any) are subtracted from the total strains increments. When the mixed u-P formulation option (*TB0PT* = MXUP on the **TB**,USER command) is used for hyperelastic material, the dStrain array is zero.
prop	Double-precision array containing the material constants defined via **TB**,USER and **TBDATA** commands. Array size is nProp. Array prop contains the material constants at current temperature point.
coords	Double-precision array containing the current coordinates of the material integration points. Array size is 3.
defGrad_t	Double-precision matrix containing deformation gradient at the beginning of the time increment. The matrix size is 3 x 3. The matrix components DefGrad_t$_{(i,j)}$ are equivalent to deformation gradient F_{ij} at the beginning of the time increment and are only available for continuum and shell elements with nonlinear deformation (**NLGEOM**,ON).
defGrad	Double-precision matrix containing current deformation gradient. The matrix size is 3 x 3. The matrix components DefGrad$_{(i,j)}$ are equivalent to deformation gradient F_{ij} at the current time and are only available for continuum and shell elements with nonlinear deformation (**NLGEOM**,ON).

UserMat Input/Output Arguments	
stress	Double-precision array containing the stresses. For time-domain analyses, its size is defined by the ncomp input value. The stress measure is the "true" stress. It is passed as the values of stresses at the beginning of the time increment and must be updated to the values of stress at the end of the time increment. For harmonic analyses, its size is stress(ncomp,2), where the first column is the real stress and the second column is the imaginary stress. For finite-deformation problems, the stresses are rotated to account for rigid body motion before they are passed in, and therefore only the co-rotational portion of stress integration is required in `UserMat`. When the mixed u-P formulation option (*TB0PT* = MXUP on the **TB**,USER command) is used for hyperelastic material, the stress is updated for deviatoric part of stress only. The calculated P is passed into usermat as stress(ncomp+1).
statev	Double-precision array containing the state variables. Its size is defined via the **TB**,STATE command. It is passed as the values of state variables at the beginning of the time increment and must be updated to the values of the state variables at the end of the time increment.
epseq	Equivalent plastic strain.
epspl	Double-precision array containing the plastic strains. The strain measure is the "true" strain. Its size is defined by the ncomp input value. It is passed as the values of the plastic strains at the beginning of the time increment and must be updated to the values of the plastic strains at the end of the time increment.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	For finite-deformation problems, the plastic strains have been rotated to account for rigid body motion before they are passed in.
sedEl	Elastic work. It is used for output purposes only and does not affect the solution.
sedPl	Plastic work. It is used for output purposes only and does not affect the solution.

UserMat Output Arguments *These values must be updated in the subroutine code.*	
keycut	Integer variable as key for loading bisection/cut control: 0 - No bisect/cut (default) 1 - Bisect/cut Set keycut = 1 when UserMat experiences convergence difficulty when solving constitutive equation integration. The bisect/cut factor is determined by the solution control. Set cutFactor to control the time stepping size. Not used in harmonic analyses.
epsZZ	Strain component at an out-of-plane direction for the plane stress state. This value is required when the thickness change is taken into account in plane stress or shell elements.
tsstif(2)	Transverse shear stiffness: Tsstif(1) - GXZ Tsstif(2) - GYZ
dsdePl	Double-precision array containing the material Jacobian matrix $\partial \Delta\sigma_{ij} / \partial \Delta\varepsilon_{ij}$. Here, the values represent the stress/strain increments, respectively. The dsdePl(i,j) value denotes the change in the i-th stress component caused by a change of the j-th strain component. By default, the program assumes that the element stiffness matrix is symmetric; therefore, you must provide a *symmetric* material Jacobian matrix *even if it is unsymmetric*. If your material requires an unsymmetric material Jacobian matrix, issue the **NROPT**,UNSYM command to define the unsymmetric stiffness matrix. When the mixed u-P formulation option (*TB0PT* = MXUP on the **TB**,USER command) is used for hyperelastic material, only the deviatoric material consistent tangent matrix is needed. For harmonic analyses, the real components are returned in dsdePl(1:ncomp,1:ncomp,1), and the imaginary components are returned in dsdePl(1:ncomp,1:ncomp,2).
pVolDer(:)	Derivatives of volumetric potential with regard to the determinant of deformation gradient: pVolDer(1) = $\partial U / \partial J$

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	pVolDer(2) = $\partial^2 U / \partial J^2$ pVolDer(3) = $\partial^3 U / \partial J^3$ This argument is needed only when a hyperelastic material is defined and mixed u-P formulation is used. For nearly incompressible hyperelastic material, a mixed u-J formulation is used. The derivatives of volume potential is to Jc, where Jc is the calculated J and is passed from strain array as strain(ncomp+1). For purely incompressible hyperelastic material, set all three derivatives pVolDer(1:3) to zero.
cutFactor	Time-step size control factor. Not used in harmonic analyses.

2.4.1.6. Table (TB) Commands for UserMat

When creating your own material model, first define the material by specifying input values for the `UserMat` subroutine (**TB**,USER). It is also necessary to specify the number of state variables used, if applicable (**TB**,STATE).

Following is more information about defining your material and specifying the number of state variables used. For detailed information about the **TB** command and arguments, see the *Command Reference*.

TB,USER Command

Issue the **TB** command using the following syntax:

TB,USER,*MAT*,*NTEMPS*,*NPTS*,*TBOPT*

where

MAT = User material ID number

NTEMPS = Number of temperature points.

NPTS = Number of material constants at a given temperature point.

TBOPT = NONLINEAR (default), LINEAR, or MXUP

The material properties at an intermediate temperature point are interpolated and passed to the `UserMat` subroutine.

Define temperatures and material constants via **TBTEMP** and **TBDATA** commands, respectively.

Example 2.1: Defining the Material for UserMat

```
tb,user,1,2,4                    ! Define material 1 as a user
                                 ! material with two temperatures
                                 ! and four data points at each
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
                                ! temperature point.
tbtemp,1.0                      ! first temp.
tbdata,1,19e5, 0.3, 1e3,100,    ! Four mat. constants for one temp.
tbtemp,2.0                      ! Second temp.
tbdata,1,21e5, 0.3, 2e3,100,    ! Four mat. constants for two temps.
```

TB,STATE Command

If you intend to use state variables with the `UserMat` subroutine, it is necessary to first specify the number of state variables. Issue the **TB** command using the following syntax:

TB,STATE,*MAT*, ,*NPTS*

where

MAT = User material ID number

NPTS = Number of state variables that you intend to use.

The command defines only the *number* of state variables and must always be associated with a user material ID. No temperatures or data are associated with the command.

By default, the program initializes state variables to zero at the beginning of an analysis. Use the **TBDATA** command to initialize your own values for state variables.

Example 2.2: Defining the Number of State Variables for UserMat

```
tb,state,1,,8                       ! Define material 1 with eight state variables
tbdata,1,c1,c2,c3,c4,c5,c6,c7,c8    ! Initialize the eight state variables.
```

2.4.1.7. Material Constitutive Integration with UserMat

The `UserMat` subroutine supports current-technology elements with all key options. However, a different material constitutive integration is necessary for the various stress states, such as general 3-D, plane stress, and beam (with or without shear-stress components).

To ensure overall numerical stability, verify that the integration scheme implemented in the subroutine is stable. The program always uses the full Newton-Raphson scheme for the global time-domain solution to achieve a better convergence rate. The material Jacobian matrix (p. 207) (dsdePl(i,j) (p. 213)) must be consistent with the material constitutive integration scheme for a better convergence rate of the overall Newton-Raphson scheme.

2.4.1.8. UserMat Restrictions

The following restrictions apply to the `UserMat` subroutine:

- The subroutine supports current-technology elements only and does not apply to legacy elements.

  For more information, see Older vs. Current Element Technologies in the *Element Reference*.

- The state variables (defined via the **TB**,STATE command) are supported only by full graphics in the POST1 postprocessor.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Because POST1 does not switch to full graphics automatically, you must issue a **/GRAPHICS**,FULL command to do so.

- The `UserMat` interface for elements with mixed u-P formulation is supported only for current-technology continuum elements (such as PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, SOLSH190, CPT212, CPT213, CPT215, CPT216, CPT217, SOLID225, SOLID226, and SOLID227), excluding plane stress state. You must set KEYOPT(6) = 1 with elements PLANE182, PLANE183, SOLID185, SOLID186, SOLID187, SOLSH190, CPT212, CPT213, CPT215, CPT216, and CPT217 and KEYOPT(11) = 1 with SOLID225, SOLID226 and SOLID227 . For elements with a plane stress assumption, a mixed u-P formulation is not needed.
- The deformation gradient defGrad(3,3) and defGrad_t(3,3) are available only for continuum and shell elements with nonlinear deformation (**NLGEOM**,ON).

2.4.1.9. Accessing Material and Element Data for UserMat

Following is the interface for accessing the material and element data :

```
*deck,get_ElmData
      subroutine get_ElmData (kchar, elemId, kMatRecPt, ncomp, vect)
c
c**********************************************************************
c
c     *** primary function
c            retrieve material record data
c            including items:
c                stress vector
c                elastic strain vector
c                plastic strain vector
c                creep   strain vector
c                thermal strain vector
c                state variables
c
c                                      ---- Guoyu Lin 4/25/2001 ----
c
c**********************************************************************
c
c *** Notice - This file contains ANSYS Confidential information ***
c
c     input arguments
c     ===============
c     kchar         (ch,sc,in)        string characters indicating
c                                     inquired quantities
c     kMatRecPt     (in,sc,in)        integration point where data
c                                     to be inquired
c     elemId        (in,sc,in)        element number
c     ncomp         (in,sc,in)        number of components to be inquired
c                                     Use the 'NCOMP' query to get the right
c                                     size for tensor quantities.
c     numWordsPt    (in,ar(*),in)     number of mat. Record
c
c     input output arguments         input desc     / output desc
c     ======================         ==========       ===========
c
c     output arguments
c     ================
c     vect          (dp,ar(*),ou)     inquired data
c
c     local variables
c     ===============
c
c**********************************************************************

c --- parameters
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c
```

The ncomp value used in `get_ElmData` (p. 216) and `put_ElmData` (p. 403) may differ from the ncomp value passed into `usermat.F` (p. 208). Use `get_ElmData` with the 'NCOMP' option to determine the correct array sizes.

For a descriptions of the input arguments and valid argument variables, see Accessing Solution and Material Data (p. 403).

2.4.1.10. Utility Functions for UserMat

The following functions are available for use with `UserMat`. Ansys, Inc. provides them for your convenience.

Utility Functions for `UserMat`	
vzero(***a*** ,*n*)	Initializes array ***a*** to zero. The value *n* is the array dimension.
vmult(***a*** , ***b*** ,*n*,*c*)	Multiplies vector ***a*** by constant c and outputs (***b*** = ***a*** * *c*) as vector ***b*** . The value *n* is the dimension for both arrays.
vmult1(***a*** ,*n*,*c*)	Multiplies vector ***a*** by constant *c* and outputs the result as itself (that is, ***a*** = ***a*** * *c*). The value *n* represents the array dimension.
maxb(***a*** , ***b*** , ***c*** , *na*, *nb*, *nc*, *n1*, *n2*, *n3*)	Multiplies two double-precision matrices and outputs the result as ***c*** (that is, ***c*** = ***a*** * ***b***). The value *na* is number of rows in matrix ***a*** , *nb* the number of rows in matrix ***b*** , and *nc* the number of rows in matrix ***c*** . The *n1* value is the number of rows in matrix ***c*** to fill, and *n2* the number of columns in matrix ***c*** to fill. The value *n3* is the number of *columns* in matrix ***a*** *and* the number of *rows* in matrix ***b*** to work with. (The number of columns in ***a*** and rows in ***b*** is the same in order to generate the inner product correctly.)

2.4.2. Subroutine UserMatTh (Creating Your Own Thermal Material Model)

`UserMatTh` is a tool for advanced users. Expertise in thermal material modeling and software programming is necessary. Developing a custom thermal material model requires validation and testing. Ansys, Inc. strongly recommends testing both single elements and multiple elements with various loading conditions to ensure correct results. `UserMatTh` supports shared memory and distributed parallel processing; however, you are responsible for ensuring that your code can use parallel processing.

The following `UserMatTh` topics are available:

2.4.2.1. UserMatTh Element Support

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.4.2.1. UserMatTh Element Support

Element support for user-defined thermal material models (**TB**,USER) is available in Material Model Element Support in the *Material Reference*.

`UserMatTh` is called at every material integration point of the elements during the solution phase.

Input values and the number of state variables (if used) for the subroutine are specified via **TB**. For more information, see Table (**TB**) Commands for UserMatTh (p. 221).

2.4.2.2. The UserMatTh API

Following is the interface for the `UserMatTh` subroutine:

```
*deck,usermatth                       USERDISTRIB
      subroutine usermatth(matId, elemId, kDomIntPt, kLayer, kSectPt,
     &                     ldstep, isubst, keycut, ncomp, nStatev,nProp,
     &                     Time, dTime, Temp, dTemp, tgrad,
     &                     ustatev, prop, coords,
     &                     dudt,dudg,flux,dfdt,dfdg,
     &                     cutFactor,hgen, dens, var1, var2, var3,
     &                     var4, var5, var6)
c*************************************************************************
c     *** primary function ***
c
c           user defined thermal material constitutive model
c
c     Attention:
c           User must define material constitutive behavior properly
c
c
c       The following demonstrates a USERMATTH subroutine for
c       a regular conductive heat transfer.
c       See "ANSYS user material subroutine USERMATTH" for detailed
c       description of how to write a USERMATTH routine.
c
c
c*************************************************************************
c Copyright ANSYS.  All Rights Reserved.
c
c     input arguments
c     ===============
c      matId     (int,sc,i)              material #
c      elemId    (int,sc,i)              element #
c      kDomIntPt (int,sc,i)              "k"th domain integration point
c      kLayer    (int,sc,i)              "k"th layer
c      kSectPt   (int,sc,i)              "k"th Section point
c      ldstep    (int,sc,i)              load step number
c      isubst    (int,sc,i)              substep number
c      ncomp     (int,sc,i)              # of components
c      nStatev   (int,sc,i)              Number of state variables
c      nProp     (int,sc,i)              Number of material constants
c
c      Temp      (dp,sc,in)              temperature at current time
c      dTemp     (dp,sc,in)              temperature increment
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c      Time      (dp,sc,in)              time at beginning of increment (t)
c      dTime     (dp,sc,in)              current time increment (dt)
c
c      prop      (dp,ar(nprop),i)        Material constants defined by TB,USER
c      coords    (dp,ar(3),i)            current coordinates
c
c      tgrad     (dp,ar(ncomp),i)        Current values of the spatial gradients of temperature
c
c
c     input output arguments
c     ======================
c      ustatev   (dp,ar(nstatev),io)     user state variables
c      hgen      (dp,sr,io)              heat generation rate per unit mass
c      dens      (dp,sr,io)              density passed in as defined mp command
c      var?      (dp,sc,io)              not used, they are reserved arguments
c                                        for further development
c
c     output arguments
c     ================
c      keycut    (int,sc,o)              loading bisect/cut control
c                                        0 - no bisect/cut
c                                        1 - bisect/cut
c                                        (factor will be determined by ANSYS solution control)
c
c      cutFactor (dp,sc,o)               time step size cut-back factor
c                                        define it if a smaller step size is wished
c                                        recommended value is 0~1
c
c      dudt      (dp,sr,o)               unit mass internal energy variation with respect to
c                                        temperature
c      dudg      (dp,ar(ncomp),o)        unit mass internal thermal energy variation  with
c                                        respect to the spatial gradients of temperature
c      flux      (dp,ar(ncomp),o)        heat flux vector
c      dfdt      (dp,ar(ncomp),o)        heat flux vector variation with respect to temperature
c      dfdg      (dp,ar(ncomp,ncomp),o)  heat flux vector variation with respect to the spatial
c                                        gradients of temperature
c
c*************************************************************************
c
c      ncomp   3   for 3D
c      ncomp   2   for 2D
c      ncomp   1   for 1D
c
c
c*************************************************************************
```

2.4.2.3. UserMatTh Variables

The `UserMatTh` subroutine uses the following Input (p. 219), Input/Output (p. 220), and Output (p. 220) variables. Do not change the Input arguments in the subroutine code.

`UserMatTh` Input Arguments	
matId	Integer variable containing the material ID number.
elemId	Integer variable containing the element number.
kDomIntPt	Integer variable containing the material integration point number.
kLayer	Integer variable containing the layer number.
kSectPt	Integer variable containing section point number.
ldstep	Integer variable containing load step number.
isubst	Integer variable containing substep number.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ncomp	Integer variable containing spatial dimension number: 1 for 1-D, 2 for 2-D, or 3 for 3-D.
nStatev	Number of state variables, specified via the *NPTS* value in the **TB**,STATE command.
nProp	Number of material constants, specified via the *NPTS* value in the **TB**,USER command.
Temp	Double-precision variable containing the current temperature.
dTemp	Double-precision variable containing the current temperature increment.
Time	Double-precision variable containing the total time at the beginning of the time increment.
dTime	Double-precision variable containing the current time increment.
prop	Double-precision array containing the material constants defined via **TB**,USER and **TBDATA** commands. Array size is nProp. Array prop contains the material constants at current temperature point.
coords	Double-precision array containing the current coordinates of the material integration points. Array size is 3.
tgrad	Double-precision array of size ncomp containing current temperature gradient.

UserMatTh Input/Output Arguments	
hgen	Double-precision variable with heat-generation information. It is passed in as the value that is predefined with the model. The value can be updated with user-defined heat generation. [1] (p. 220)
ustatev	Double-precision array containing the state variables. Its size is defined via the **TB**,STATE command. It is passed as the values of state variables at the beginning of the time increment and must be updated to the values of the state variables at the end of the time increment.[1] (p. 220)
dens	Double precision variable with density information. It is passed in as the value that is predefined with the model. The value can be updated with user-defined density.[1] (p. 220)

1. In a coupled structural-thermal analysis using SOLID225, **SOLID226** and SOLID227:

   - the heat generation rate (hgen) is interpreted as heat flow rate per unit volume,
   - the state variables (ustatev) cannot be updated with user-defined values in `UserMatTh`,
   - the density (dens) cannot be updated with user-defined value in `UserMatTh`.

UserMatTh Output Arguments *These values must be updated in the subroutine code.*	
keycut	Integer variable as key for loading bisection/cut control: 0 - No bisect/cut (default) 1 - Bisect/cut Set keycut = 1 when `UserMatTh` experiences convergence difficulty when solving constitutive equation integration. The bisect/cut factor is determined by the solution control.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

cutFactor	Double-precision variable containing the user-defined time-step size-control factor.
dudt	Double-precision variable containing the unit mass internal energy variation with respect to temperature.
dudg	Double-precision array of size (ncomp,ncomp) containing the unit mass internal thermal energy variation with respect to the spatial gradients of temperature.
flux	Double-precision array of size ncomp containing the heat flux vector.
dfdt	Double-precision array of size ncomp containing the heat flux vector variation with respect to temperature.
dfdg	Double-precision array of size (ncomp,ncomp) containing the heat flux vector variation with respect to the spatial gradients of temperature.

2.4.2.4. Table (**TB**) Commands for UserMatTh

When creating your own material model, first define the material by specifying input values for the `UserMatTh` subroutine (**TB**,USER). It is also necessary to specify the number of state variables used, if applicable (**TB**,STATE).

In a coupled-field analysis with structural and thermal degrees of freedom, use **TB**,USER with *TBOPT* = THERM to define the thermal material independently of the structural material model. The THERM option is available with elements SOLID225, SOLID226 and SOLID227.

Following is more information about defining your material and specifying the number of state variables used. For detailed information about the **TB** command and arguments, see the *Command Reference*.

TB,USER Command

Issue the **TB** command using the following syntax:

TB,USER,*MAT*,*NTEMPS*,*NPTS*,*TBOPT*

where

MAT = User material ID number

NTEMPS = Number of temperature points.

NPTS = Number of material constants at a given temperature point.

TBOPT = NONLINEAR (default), LINEAR, or THERM

The material properties at an intermediate temperature point are interpolated and passed to the `UserMatTh` subroutine.

Define temperatures and material constants via **TBTEMP** and **TBDATA** commands, respectively.

Example 2.3: Defining the Material for UserMatTh

```
tb,user,1,2,4                  ! Define material 1 as a user
                               ! material with two temperatures
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
                                ! and four data points at each
                                ! temperature point
tbtemp,1.0                      ! first temperature
tbdata,1,19e5,0.3,1e3,100,      ! four material constants for
                                ! one temperature
tbtemp,2.0                      ! Second temperature
tbdata,1,21e5,0.3,2e3,100,      ! Four material constants for
                                ! two temperatures
```

For an example of `UserMatTh` in a coupled-field analysis, see Appendix D: Structural-Thermal User Material (UserMat, UserMatTh) Example (p. 405).

TB,STATE Command

If you intend to use state variables with the `UserMatTh` subroutine, it is necessary to first specify the number of state variables. Issue the **TB** command using the following syntax:

TB,STATE,*MAT*, ,*NPTS*

where

MAT = User material ID number.

NPTS = Number of state variables that you intend to use.

The command defines only the *number* of state variables and must always be associated with a user material ID. No temperatures or data are associated with the command.

By default, the program initializes state variables to zero at the beginning of an analysis. Use the **TBDATA** command to initialize your own values for state variables.

2.4.2.5. UserMatTh Restrictions

The following restrictions apply to the `UserMatTh` subroutine:

- The subroutine supports current-technology elements only and does not apply to legacy elements. For more information, see UserMatTh Element Support (p. 218).
- The state variables (defined via the **TB**,STATE command) are supported only by full graphics in the POST1 postprocessor.

  Because POST1 does not switch to full graphics automatically, you must issue a **/GRAPHICS**,FULL command to do so.
- Element FLUID116 is supported only when KEYOPT(1) = 1. For this element, you can use `UserMatTh` to specify material properties KXX, C, and DENS only.

2.4.2.6. Utility Functions for UserMatTh

The following functions are available for use with `UserMatTh`. Ansys, Inc. provides them for your convenience.

Utility Functions for `UserMatTh`

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

vzero(***a*** ,*n*)	Initializes array ***a*** to zero. The value *n* is the array dimension.
vmult(***a*** , ***b*** ,*n*,*c*)	Multiplies vector ***a*** by constant c and outputs (***b*** = ***a*** * *c*) as vector ***b*** . The value *n* is the dimension for both arrays.
vmult1(***a*** ,*n*,*c*)	Multiplies vector ***a*** by constant *c* and outputs the result as itself (that is, ***a*** = ***a*** * *c*). The value *n* represents the array dimension.
maxb(***a*** , ***b*** , ***c*** , *na*, *nb*, *nc*, *n1*, *n2*, *n3*)	Multiplies two double-precision matrices and outputs the result as ***c*** (that is, ***c*** = ***a*** * ***b***). The value *na* is number of rows in matrix ***a*** , *nb* the number of rows in matrix ***b*** , and *nc* the number of rows in matrix ***c*** . The *n1* value is the number of rows in matrix ***c*** to fill, and *n2* the number of columns in matrix ***c*** to fill. The value *n3* is the number of *columns* in matrix ***a*** *and* the number of *rows* in matrix ***b*** to work with. (The number of columns in ***a*** and rows in ***b*** is the same in order to generate the inner product correctly.)

2.4.3. Subroutine UserHyper (Writing Your Own Isotropic Hyperelasticity Laws)

Use the subroutine `UserHyper` when you issue **TB**,HYPER with *TB0PT* = USER.

```
*deck,UserHyper                       USERDISTRIB
c Copyright ANSYS.  All Rights Reserved.
      subroutine UserHyper(
     &                     prophy, incomp, nprophy, invar,
     &                     potential, pInvDer)
c*********************************************************************
c
c     *** Example of user hyperelastic routine
c
c           This example uses Arruda hyperelasticity model
c           which is the same ANSYS TB,HYPER,,,,BOYCE
c
c     input arguments
c     ===============
c      prophy       (dp,ar(*),i)     material property array
c      nprophy     (int,sc,i)        # of material constants
c      invar       dp,ar(3)          invariants
c
c     output arguments
c     ================
c      incomp      (log,sc,i)         fully incompressible or compressible
c      potential    dp,sc             value of potential
c      pInvDer       dp,ar(10)        der of potential wrt i1,i2,j
c                                      1 - der of potential wrt i1
c                                      2 - der of potential wrt i2
c                                      3 - der of potential wrt i1i1
c                                      4 - der of potential wrt i1i2
c                                      5 - der of potential wrt i2i2
c                                      6 - der of potential wrt i1j
c                                      7 - der of potential wrt i2j
```

```
c                                       8 - der of potential wrt j
c                                       9 - der of potential wrt jj
c
c**********************************************************************
c
c --- parameters
c
```

2.4.4. Subroutine UserHyperAniso (Writing Your Own Anisotropic Hyperelasticity Laws)

The `UserHyperAniso` subroutine defines the potential derivatives for a strain-energy potential that is a function of isochoric strain invariants and anisotropic fiber invariants.

2.4.4.1. Input Parameters

Use the `UserHyperAniso` subroutine when you issue **TB**,AHYPER with *TBOPT* = USER.

Issue **TBDATA** to specify the invariant set type (SetType). The only valid set type is 101, which defines the invariant set as the isochoric strain invariants and isochoric fiber invariants defined below.

Example 2.4: User-Defined Anisotropic Hyperelastic Material Model

```
/prep7
TB,AHYPER,1,,,USER
TBDATA,1,101        ! define the invariant set type
```

You can define an optional set number (SetNumber) in a data table initiated via *TBOPT* = UNUM. This value is passed into the subroutine for use in distinguishing between user-defined material behaviors.

Example 2.5: User-Defined Anisotropic Hyperelastic Set Number

```
/prep7
TB,AHYPER,1,,,UNUM
TBDATA,1,1          ! define the invariant set number
```

Define the material parameters in a table initiated via *TBOPT* = AU01.

Example 2.6: User-Defined Anisotropic Hyperelastic Parameters

```
/prep7
TB,AHYPER,1,,,AU01
 !a1, a2, a3
    tbdata,1,a1
 !b1, b2, b3
    tbdata,4,b1
 !c1, c2, c3,
    tbdata,7,c1, c2, c3,
 !d2, d3, d4, d5, d6
    tbdata,12,0,0.0,0.0,0.0,0.0
 !e1, e2, e3
    tbdata,17,e1, e2, e3
 !f1, f2, f3, f4, f5, f6
    tbdata,22,0,0.0,0.0,0.0,0.0
 !g1, g2, g3, g4, g5, g6
    tbdata,27,0,0.0,0.0,0.0,0.0
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
!d
   tbdata,31,1e-5
```

Specify the fiber directions in a data table with *TB0PT* = FB01. Three values define the direction of a fiber. The direction is relative to the element coordinate system and should have a magnitude equal to 1.0. The number of fibers (25 maximum) is determined from the number of defined values.

Example 2.7: User-Defined Anisotropic Hyperelastic Fiber Directions

```
/prep7
TB,AHYPER,1,,,FB01
TBDATA,1,1,0,0
TBDATA,4,1/sqrt(2),1/sqrt(2),0
```

2.4.4.2. Invariants and Potential Derivatives

Define the strain-energy potential as a function of the isochoric strain invariants and the anisotropic invariants that depend on the fiber directions and deformation.

$$W=W\left(\bar{I}_1,\bar{I}_2,J,A_1,\ldots,A_n\right)$$

where $\bar{I}_1,\bar{I}_2,J$ are the isochoric strain invariants and $A_1,\ldots,A_n$ are the fiber directions. The strain-energy potential depends on isochoric fiber invariants that have the form:

$$\bar{I}_4=A_i\cdot\bar{C}A_i$$

$$\bar{I}_5=A_i\cdot\bar{C}^2A_i$$

$$\bar{I}_6=A_i\cdot\bar{C}A_j,\quad i\neq j$$

where $\bar{I}_4$ is a first-order fiber invariant, $\bar{I}_5$ is a second-order fiber invariant, and $\bar{I}_6$ is a mixed-fiber variant.

Mixed second-order fiber invariants are not used in the strain-energy potential definition.

The isochoric strain invariants are numbered 1, 2, and 3 respectively in the Invar(*) array. The fiber-invariant numbering scheme is obtained from the following function:

ni = FIindx(SetType,InvType,Fib1,Fib2)

where:

ni = returned value of invariant number in Invar(*) array

SetType = set type passed into `UserHyperAniso`

InvType = invariant type: 1 = first-order, 2 = second-order

Fib1 = first fiber number, *i*

Fib2 = second fiber number, *i* or *j*

The `UserHyperAniso` subroutine should calculate and return the first and second derivatives of the user-defined strain-energy potential with respect to the invariants. For mixed u-P formulations

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

that are not incompressible, the third-order invariant with respect to J is also required. (Third-order derivatives of the strain-energy potential with respect to other invariants are not required.)

The compressibility of the material is determined from the second derivative of the strain-energy potential with respect to J in the reference configuration during the first solution step. If this derivative is zero, the material is incompressible and the appropriate mixed u-P formulation is used. The material should not change between compressible and incompressible behavior during the simulation.

The strain-energy-potential derivatives are stored in the arrays pD1(*), pD2(*) and pD3(*) via the following subroutine:

put_PDer(SetType,nFib,Order,In1,In2,In3,pD1,pD2,pD3,val)

where:

SetType = set type passed into UserHyperAniso

nFib = number of fibers passed into UserHyperAniso

Order = derivative order to be stored (1, 2, or 3)

In1, In2, In3 = invariant numbers corresponding to the respective derivatives

pD1, pD2, pD3 = potential derivative arrays passed into UserHyperAniso

val = value of the derivative that is added to the value in the array

2.4.4.3. UserHyperAniso API

```
*deck,UserHyperAniso                      USERDISTRIB
      subroutine UserHyperAniso(SetType,SetNumber,incomp,upkey,nprophy,
     &                    prophy,nFib,fibDir,ninv,Invar,potential,
     &                    pD1,pD2,pD3)
c***********************************************************************
c
c     *** Example of user anisotropic hyperelastic routine
c
c           This example reproduces the Polynomial hyperelasticity
c          model which is the same as TB,AHYPER,,,,POLY
c
c     input arguments
c     ===============
c      SetType     (int,sc,i)        Type of invariant set
c      SetNumber   (int,sc,i)        User input invariant set number
c      incomp      (log,sc,i)        incompressibility flag (.true. for incompressible)
c      upkey       (int,sc,i)        mixed uP key (/=0 for mixed uP formulation)
c      nprophy     (int,sc,i)        number of prophy values
c      prophy      (dp,ar(*),i)      material property array
c      nFib        (int,sc,i)        number of fibers
c      invar       dp,ar(3)          invariants
c      fibDir      (dp,ar(3,*),i)    original fiber directions array
c      ninv        (int,sc,i)        number of invariants
c      Invar       (dp,ar(*),i)      set of Invariants
c
c     output arguments
c     ================
c      potential    dp,sc             value of potential
c      pD1       dp,ar(*)             1st derivatives of potential wrt Invar(*)
c      pD2       dp,ar(*)             2nd derivatives of potential wrt Invar(*)
c      pD3       dp,ar(*)             3rd derivatives of potential wrt J
c
c***********************************************************************
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.4.5. Subroutine UserCreep (Defining Creep Material Behavior)

Use the subroutine `UserCreep` to define creep material behavior. The subroutine is applicable when you issue the **TB**,CREEP, and with *TBOPT* = 100.

`UserCreep` supports shared memory and distributed parallel processing; however, you are responsible for ensuring that your code can use parallel processing.

The subroutine is called at all integration points of elements for which the material is defined by this command. The program always uses implicit time integration for this creep option. You can use plasticity options (BISO, BKIN, NLISO, PLASTIC) to define the plastic behavior of materials. Creep and plastic strain are calculated simultaneously when both creep and plasticity are defined for a material.

Using this subroutine, you can specify a "uniaxial" creep law that is generalized to the multi-axial state via the program's general time-dependent viscoplastic material formulation. You can use and update internal state variables in the subroutine. The number of state variables must be defined (**TB**,STATE).

Please see the **TB** command description for more information.

```
*deck,usercreep                       USERDISTRIB
      SUBROUTINE usercreep (impflg, ldstep, isubst, matId , elemId,
     &                      kDInPt, kLayer, kSecPt, nstatv, nprop,
     &                      prop  , time  , dtime , temp  , dtemp ,
     &                      toffst, Ustatev, creqv , pres  , seqv  ,
     &                      delcr , dcrda)
c**********************************************************************
c     *** primary function ***
c           Define creep laws when creep table options are
c           TB,CREEP with TBOPT=100.
c           Demonstrate how to implement usercreep subroutine
c
c            Creep equation is
c               dotcreq := k0 * seqv ^ n * creqv ^ m * exp (-b/T)
c
c               seqv  is equivalent effective stress (Von-Mises stress)
c               creqv is accumulated equivalent creep strain
c               T     is the temperature
c               k0, m, n, b are materials constants,
c
c           This model corresponds to  primary creep function  TBOPT = 1
c
c                                                          gal 10.01.1998
c
c**********************************************************************
c Copyright ANSYS.  All Rights Reserved.
c
c     input arguments
c     ===============
c      impflg   (in ,sc   ,i)            Explicit/implicit integration
c                                        flag (currently not used)
c      ldstep   (in ,sc   ,i)            Current load step
c      isubst   (in ,sc   ,i)            Current sub step
c      matId    (in ,sc   ,i)            number of material index
c      elemId   (in ,sc   ,i)            Element number
c      kDInPt   (in ,sc   ,i)            Material integration point
c      kLayer   (in ,sc   ,i)            Layer number
c      kSecPt   (in ,sc   ,i)            Section point
c      nstatv   (in ,sc   ,i)            Number of state variables
```

```
c      nprop    (in ,sc   ,i)            size of mat properties array
c
c      prop     (dp ,ar(*),i)            mat properties array
c                                        This array is passed all the creep
c                                        constants defined by command
c                                        TBDATA associated with TB,CREEP
c                                        (do not use prop(13), as it is used
c                                        elsewhere)
c                                        at temperature temp.
c      time                              Current time
c      dtime                             Current time increment
c      temp                              Current temperature
c      dtemp                             Current temperature increment
c      toffst   (dp, sc,   i)           temperature offset from absolute zero
c      seqv     (dp ,sc  , i)           equivalent effective stress
c      creqv    (dp ,sc  , i)           accumulated equivalent creep strain
c      pres     (dp ,sc  , i)           hydrostatic pressure stress, -(Sxx+Syy+Szz)/3
c                                        note that: constitutive consistency is not accounted for
c                                        if creep strains are function of pressure
c
c     input output arguments             input desc     / output desc
c     =====================              ==========       ===========
c      Ustatev  (dp,ar(*), i/o)          user defined iinternal state variables at
c                                        time 't' / 't+dt'.
c                                        This array will be passed in containing the
c                                        values of these variables at start of the
c                                        time increment. They must be updated in this
c                                        subroutine to their values at the end of
c                                        time increment, if any of these internal
c                                        state variables are associated with the
c                                        creep behavior.
c
c     output arguments
c     ================
c      delcr    (dp ,sc  , o)            incremental creep strain
c      dcrda    (dp,ar(*), o)            output array
c                                        dcrda(1) - derivitive of incremental creep
c                                                   strain to effective stress
c                                        dcrda(2) - derivitive of incremental creep
c                                                   strain to creep strain
c
c     local variables
c     ===============
c      c1,c2,c3,c4 (dp, sc, l)           temporary variables as creep constants
c      expt        (dp ,sc, l)           temporary variable
c      t           (dp ,sc, l)           temporary variable
c
c**********************************************************************
c
c --- parameters
c
```

2.4.6. Subroutine user_tbelastic (Defining Material Linear Elastic Properties)

Subroutine `user_tbelastic` provides an interface for defining your own material linear elastic properties (**TB**,ELASTIC). The following topics are available:

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.4.6.1. Overview of the user_tbelastic Subroutine

The `user_tbelastic` subroutine can define material linear elastic properties as a function of temperature or coordinates. The subroutine is called at the material integration points of elements for which the definition of material elastic properties is a user option. The material properties defined are based on the material coordinate system of the elements.

You can use the subroutine with most current-technology elements and with most nonlinear material models.

For more information about these material models, see the documentation for the **TB** command in the *Command Reference*.

2.4.6.2. Data Types Supported by user_tbelastic

The `user_tbelastic` subroutine can define the following types of material property data:

- **Isotropic elasticity with two constants**

  Define the Young's modulus (EX) and Poisson's ratio (NUXY) material constants

- **General orthotropic elasticity with nine constants**

  Define the normal modulus, shear modulus, and minor Poisson's ratios. The order is as follows: EX, EY, EZ, GXY, GXZ, GYZ, NUXY, NUXZ, NUYZ. All nine constants must be defined; no default values are assigned.

- **Anisotropic elasticity with 21 constants**

  Define the material elastic stiffness matrix. The matrix consists of 21 constants, and all must be defined.

2.4.6.3. Table (TB) Command for user_tbelastic

Issue a **TB** command using the following syntax to access the `user_tbelastic` subroutine interface:

TB,ELASTIC,*mat*,,*npts*,USER

The ELASTIC argument accesses the elastic material property data table. (For more information, see the documentation for the **TB** command's ELASTIC option in the *Command Reference*.)

The *mat* value represents the material number, and the *npts* value is the number of material constants.

The USER argument accesses the interface to the `user_tbelastic` subroutine.

2.4.6.4. User Interface for user_tbelastic

The `user_tbelastic` interface consists of six arguments, as follows:

- Four input arguments for the element number, material number, coordinate array, and temperature

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- One input/output argument for the number of material constants
- One output argument consisting of the material constants array

The syntax is as follows: **`SUBROUTINE user_tbelastic(elemId, matId, coords, temp, nprop, prop)`**

Argument	Input (I) *or* Output (O)	Definition
`elemId`	I	Element number
`matId`	I	Material number
`coords`	I	Coordinates of material integration point at initial configuration (geometry)
`temp`	I	Current temperature at material integration point
`nprop`	I / O	Number of constants to be returned (input) or actually returned (output), as follows: 2 - isotropic elasticity 9 - orthotropic elasticity 21 - anisotropic elasticity The value for this argument is obtained via the **TB**,ELASTIC command, and is passed into the subroutine. However, you can redefine this value in the subroutine, which then returns it.
`prop`	O	The material elastic constants to be defined

2.4.6.5. The user_tbelastic API

Following is the interface to the `user_tbelastic` subroutine:

```
*deck,user_tbelastic                      USERDISTRIB
      SUBROUTINE user_tbelastic(elemId, matId, coords, temp,
     &                          nprop,  prop)
c**********************************************************************
c     *** primary function ***
c           user interface for elastic material constants
c
c**********************************************************************
c Copyright ANSYS.  All Rights Reserved.
c
c     input arguments
c     ===============
c      elemId   (in, sc   , i)    Element number
c      matId    (in, sc   , i)    Number of material index
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c      temp     (dp, sc   , i)    Current temperature
c
c      coords   (dp, ar(5), i)    Coordinates at initial configuration
c                                  For continuum elements:
c                                   1-3:  coordinates of integration point
c                                   4-5:  not used
c                                  For line elements:
c                                   1-3:  coordinates of integration point
c                                            along line member axis
c                                   4-5:  offsets in element y and z directions
c
c      output arguments
c     ======================
c      nprop    (in, sc   , o)    Number of constants
c                                    2  - isotropic   elasticity
c                                    9  - orthotropic elasticity
c                                    21 - anisotropic elasticity
c      prop     (dp, ar(*), o)    Material elastic constants (stiffness)

c     local variables
c     ===============
c
c**********************************************************************
c
c --- parameters
c
```

2.4.6.6. Usage Example for user_tbelastic

In this example, three elements in parallel are subjected to uniaxial tension.

Element 1 is a SOLID185 element defined via the **MP** command with linear isotropic elasticity.

Element 2 is a SOLID185 element defined via the user-defined elastic material properties interface.

Element 3 is a SHELL181 element defined via the user-defined elastic material properties interface.

Solid elements are a unit cubic with a 1 mm edge. The shell element is a unit square with a 1 mm edge. The Young's modulus is 210e3 MPa, and the Poisson's ratio is 0.3.

Example Input

```
/batch
/com
/com example for user elastic material property interface
/com
/com element 1 solid185 defined via standard MP command
/com element 2 solid185 defined using ansys elastic material interface
/com element 3 shell181 defined using ansys elastic material interface
/com
/prep7

esize,,1
et,1,185
et,2,181

mp,ex,1,210e3
mp,nuxy,1,0.3
tb,elastic,2,1,2,user   ! user-defined elastic material interface

! SOLID185 element
mat,2
block,,1,,1,,1
vmesh,1
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
mat,1
block,,1,,1,,1
vmesh,2

! SHELL181 element
sectype,1,shell
secdata, 0.100000,1
secdata, 0.100000,2
rect,,1,,1
secn,1
mat,2
type,2
amesh,1

elist,all,all

nsel,s,loc,x
d,all,ux
nsel,s,loc,y
d,all,uy
nsel,s,loc,z
d,all,uz

/solu

nsel,s,loc,x,1
d,all,ux,0.05
alls
solve

fini
/post1
set,1
pres,s
pres,epel
fini
```

2.4.7. Subroutine userfc (Defining Your Own Failure Criteria)

```
*deck,userfc                      USERDISTRIB
      subroutine userfc (elem,matlay,iott,tem,elim,slim,
     x                   eps, sig, nfcOut, fc)
c     primary function: user subroutine for defining your own failure criterion
c *** secondary functions: none
c
c *** user programmable functions may not be used in parallel processing ***
c     this is currently only available with
c
c         *** Copyright ANSYS.  All Rights Reserved.
c         *** ansys, inc.
c
c *** Notice - This file contains ANSYS Confidential information ***
c
c  input arguments:
c     variable (typ,siz,intent)    description
c     elem     (int,sc,in)       - element number
c     elim     (dp,ar(9),in)     - failure strains at the current temperature
c                                    (see FC command input with Lab1 = EPEL)
c     slim     (dp,ar(12),in)    - failure stresses and coupling coefficients
c                                    at the current temperature
c                                    (see FC command input with Lab1 = S)
c     eps      (dp,ar(6),in)     - vector of strains
c     sig      (dp,ar(6),in)     - vector of stresses
c     tem      (dp,sc,in)        - temperature at this point in the model
c     matlay   (int,sc,in)       - material number
c     iott     (int,sc,in)       - unit number for writing
c
c  output arguments:
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c     variable (typ,siz,intent)    description
c     nfcOut   (int,sc, out)      - number of user fc computed
c     fc       (dp,ar(9),out)    - user failure criterion
c
```

2.4.8. Subroutine userCZM (Defining Your Own Cohesive Zone Material)

Define your own interfacial cohesive material law via the **TB**,CZM,,,,USER command.

Issue the **TBDATA** command to define the material constants. Data may be temperature-dependent and is interpolated at the current temperature of the material integration point and passed to the subroutine.

For more information, see User-Defined Cohesive Material (UserCZM) and Using State Variables with the UserCZM Subroutine in the *Material Reference*.

Following is the user cohesive material interface:

```
*deck,userCZM                       USERDISTRIB
      subroutine userCZM (matId, elemId, kMatIntPt, ldstep,isubst,
     &                    keycut, ncomp,nProp, nstatev,
     &                    Time, dTime, Temp, dTemp,
     &                    coords, prop, Strain, dStrain,
     &                    stress, dsdePl, sedEl, sedPl, statev,
     &                    var1, var2, var3, var4, var5)
c
c**********************************************************************
c
c     *** primary function ***
c
c           user cohesive zone model example
c
c           Commands
c             TB,CZM,mat,NTEMP,NPTS,user
c                TBTEMP if mat. constants are temperature dependent
c                TBDATA define material constants
c
c**********************************************************************
c
c     input arguments
c     ===============
c      matId     (int,sc,in)             material #
c      elemId    (int,sc,in)             element #
c      kMatIntPt (int,sc,in)             material integration point #
c      ldstep    (int,sc,in)             load step number
c      isubst    (int,sc,in)             substep number
c      ncomp     (int,sc,in)             number of stress, strain components
c      nProp     (int,sc,in)             Number of material ocnstants
c      nstatev   (int,sc,in)             Number of state variables
c
c      Temp      (dp ,sc,in)             temperature at beginning of time increment
c      dTemp     (dp ,sc,in)             temperature increment
c      Time      (dp ,sc,in)             time at beginning of increment (t)
c      dTime     (dp ,sc,in)             time increment (dt)
c
c      prop     (dp,ar(nprop),i)         Material constants defined by TB command
c      Strain   (dp,ar(ncomp),i)         Interface separation at beginning of time increment
c      dStrain  (dp,ar(ncomp),i)         Interface separation increment
c      coords   (dp,ar(3),i)             current coordinates
c
c     output arguments
c     ================
c      stress   (dp,ar(nTesn),io)        Traction stress
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c      sedEl    (dp,sc,io)                elastic work
c      sedPl    (dp,sc,io)                plastic work
c      keycut   (int,sc,io)               loading bisect/cut control
c                                         0 - no bisect/cut
c                                         1 - bisect/cut
c                                         (factor will be determined by ANSYS solution control)
c      dsdePl   (dp,ar(ncomp,ncomp),io)   consistent tangent jacobian matrix
c
c     input output arguments
c     ======================
c      statev   (dp,ar(nstatev,io)        user defined solution state variables
c
c     misc.
c     ======================
c      var1, var2, var3, var4, var5       currently not used
c
c     local variables
c     ======================
c
c      debugflag (in,sc, l)               debugflag to print debug information
c
c
c*************************************************************************
c
```

2.4.9. Subroutine userswstrain (Defining Your Own Swelling Laws)

You can define your own swelling strain option via the **TB**,SWELL,,,,USER command.

Use the **TBDATA** command to define the material constants. Data may be temperature-dependent and is interpolated at the current temperature of the material integration point and passed to the subroutine.

For more information, see Swelling in the *Material Reference*.

```
*deck,userswstrain                        USERDISTRIB
      subroutine userswstrain  (elemId,  kMatPoint
     &,                         matId,   nprop,   propv
     &,                         time,    dtime
     &,                         efvs,    defv
     &,                         sweqt,   dsweq
     &,                         swvi,    swvo)
c
c
c *** primary function:    compute user defined swelling strain
c
c *** Notice - This file contains ANSYS Confidential information ***
c Copyright ANSYS.  All Rights Reserved.
c
c  input arguments:
c     matId    (int,sc,in)       - material #
c     elemId   (int,sc,in)       - element  #
c     kMatPoint(int,sc,in)       - element integration point #
c     nprop    (int,sc,in)       - number of material constants
c     time     (int,sc,in)       - current time
c     dtime    (int,sc,in)       - current time increment
c     propv    (dp,ar(nprop),in) - array of material constants
c                                  (the data input via TBDATA command)
c     efvs     (dp,ar(*)    ,in) - field variables array
c                                  efvs(1) - current temperature
c                                  efvs(2) - current fluence
c     defv     (dp,ar(*)    ,in) - incremental field variables array
c                                  defv(1) - temperature increment
c                                  defv(2) - fluence increment
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c     sweqt    (dp,sc       ,in) - equivalent swelling strain at time t
c
c  output arguments:
c     dsweq    (dp,sc       ,in) - incremental equivalent swelling strain
c  not used arguments:
c     swvi     (dp,sc       ,in) - not currently used
c     swvo     (dp,sc       ,in) - not currently used
c
c*************************************************************
c
```

2.4.10. Subroutine userck (Checking User-Defined Material Data)

```
*deck,userck                      USERDISTRIB
      subroutine userck (curmat,ntb,tb)
c *** primary function:    check the user-defined material data,
c                          input with the TB,user command.
c *** secondary functions:  none
c
c         *** Copyright ANSYS.  All Rights Reserved.
c         *** ansys, inc.
c *** Notice - This file contains ANSYS Confidential information ***
c
c  input arguments:
c     curmat   (int,sc,in)       - current material number
c     ntb      (int,sc,in)       - dimension of tb
c     tb       (dp,ar(ntb),in)   - input table
c
c  output arguments:
c     none
c
```

2.4.11. Supporting Function egen

The function egen (*kcomp,ep,nuxy*) (*function*) combines *kcomp* strain components (*ep*) per:

```
*deck,egen
      function egen (kcomp,ep,posn)
c primary function:    combines strain components to give an "overall" strain
c                      used in creep and plasticity calculations
c secondary functions: none

c    formulation of overall value is by:

c          ____________________________________________________________
c         /1             2             2             2  1    2      2      2
c    \  / -*((ep  - ep ) + (ep  - ep ) + (ep  - ep ) + -*(ep   + ep   + ep ))
c     \/  2     1     2       2     3       3     1    2   4      5      6
c    ------------------------------------------------------------------------
c                               (1 + posn)

c *** Notice - This file contains ANSYS Confidential information ***

c     typ=int,dp,log,chr,dcp   siz=sc,ar(n)   intent=in,out,inout

c  input arguments:
c     variable (typ,siz,intent)    description
c     kcomp    (int,sc,in)       - number of components of strain
c     ep       (dp,ar(6),in)     - the strain components
c     posn     (dp,sc,in)        - poisson's ratio

c  output arguments:
```

```
c     egen     (dp,func,out)     - the combined strain value
c
```

2.4.12. Subroutine userfld (Update User-Defined Field Variables)

You can create your own field variables using via the **INISTATE** command and supported materials (**TB**). Node-based initial state helps to initialize the user-defined field variables that are then used by the **TB** database to evaluate the material properties at an integration point. `userfld` allows you to examine the current state at the integration point and to modify the field variables as needed.

For more information, see Understanding Field Variables in the *Material Reference*.

```
*deck,userfld                       USERDISTRIB
      subroutine userfld
     &          ( matId, elemId,
     &            ldstep, isubst, time, dtime,
     &            kDomIntPt, kLayer, kSectPt,
     &            nDirect, nShear, nComp, nStatev,
     &            coords,
     &            Temp,dTemp )
c**********************************************************************
c     *** primary function ***
c
c           Edit Field Variables During Solution
c
c      Attention:
c
c**********************************************************************
c Copyright ANSYS.  All Rights Reserved.
c
c     input arguments
c     ===============
c      matId     (int,sc,i)              material #
c      elemId    (int,sc,i)              element #
c      ldstep    (int,sc,i)              load step num
c      isubst    (int,sc,i)              substep num
c      time      (int,sc,d)              time
c      dtime     (int,sc,d)              time inc
c      kDomIntPt (int,sc,i)              "k"th domain integration point
c      kLayer    (int,sc,i)              "k"th layer
c      kSectPt   (int,sc,i)              "k"th Section point
c      nDirect   (int,sc,in)             # of direct components
c      nShear    (int,sc,in)             # of shear components
c      ncomp     (int,sc,in)             nDirect + nShear
c      nstatev   (int,sc,i)              Number of state variables
c      Temp      (dp,sc,in)              temperature at beginning of
c                                        time increment
c      dTemp     (dp,sc,in)              temperature increment
c      coords   (dp,ar(3),i)             current coordinates
c
c     input output arguments
c     ======================
c
c
c     output arguments
c     ================
c
c**********************************************************************
c                        List of Supported Field Variable Types
c**********************************************************************
c
c     FLD_USER_1_TYPE
c     FLD_USER_2_TYPE
c     FLD_USER_3_TYPE
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c     FLD_USER_4_TYPE
c     FLD_USER_5_TYPE
c     FLD_USER_6_TYPE
c     FLD_USER_7_TYPE
c     FLD_USER_8_TYPE
c     FLD_USER_9_TYPE
c
c*************************************************************************
```

2.4.13. Subroutine userthstrain (Defining Your Own Thermal Strain)

You can define the thermal strain via the **TB**,CTE,,,,USER command and the `userthstrain` subroutine.

Issue the **TBDATA** command to define the material constants. Data can be field-dependent, and is interpolated at the current field values of the material integration point and passed to the subroutine.

For more information, see Thermal Expansion in the *Material Reference*.

```
*deck,userthstrain                      USERDISTRIB
      subroutine userthstrain (nprop, propv,
     &                         ncomp, epth)
c
c *** primary function:    compute the thermal strain
c
c *** Notice - This file contains ANSYS Confidential information ***
c
c  input arguments:
c     nprop    (int,sc,in)       - number of material constants
c     propv    (dp,ar(ncomp),in) - array of material constants
c                                  e.g. coefficients of thermal
c                                       expansion in x,y,z order
c     ncomp    (int,sc,in)       - number of strain components
c                                  6  for 3-d elements   (x,y,z,xy,yz,xz)
c                                  4  for plane elements (x,y,z,xy)
c                                  3  for beam elements  (x,xy,xz)
c                                  1  for line elements  (x)
c
c  output arguments:
c     epth     (dp,ar(ncomp),out) - thermal strains
c
```

2.5. Subroutines for Customizing Contact Interfacial Behavior

The following subroutines enable you to customize contact interfacial behavior for contact elements (CONTA172, CONTA174, CONTA175, CONTA177, and CONTA178):

The subroutines enable you to:

- Perform a user-defined operation on real constants (subroutine `usercnprop` (p. 238))
- Write your own friction laws (subroutine `userfric` (p. 242))

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Write your own contact interactions (subroutine `userinter` (p. 244))
- Write your own wear law (subroutine `userwear` (p. 250))

2.5.1. Subroutine usercnprop (Programming Your Own Contact Properties)

This subroutine applies to the CONTA17*x* contact elements.

```
*deck,usercnprop                        USERDISTRIB
      subroutine usercnprop (ndim,coor,nkeyopt,keyopt,nrl,rlconst,
     x  nintIn,intIn,nrealIn,realIn,kupdhis,localr,nuval,nintp,usvr,
     x  ncomp,stress,strain0,strain,kstat,mu,kcnprop,cnprop,keyerr)
c
c *** primary function:   Allow users to define their own contact properties
c                         in real constant table
c                         This logic is accessed with real constant defined
c                         by table name: %_CNPROP%
c                         (e.g. rmod,cid,kcnprop,%_CNPROP%)
c
c *** Notice - This file contains ANSYS Confidential information ***
c
c
c         Copyright ANSYS.  All Rights Reserved.
c         *** ansys, inc.
c
c  input arguments:
c     variable (type,sze,intent)    description
c
c     elem     (int,sc,in)         - element number
c     intpt    (int,sc,in)         - element integration point number
c     ndim     (int,sc,in)         - number of dimensions of the problem
c                                    = 2 2D
c                                    = 3 3D
c     nintp    (int,sc,in)         - the total number of integration points of
c                                    an element to be used with this routine
c     nuval    (int)               - number of additional state variables per
c                                    integration point
c         note:  nuval x nintp = nstv(on nsvr command); cannot exceed 840!
c
c     intIn    (int,ar(*),in)      - integer variables passed in
c     intIn    (int,ar(*),in)      - integer variables passed in
c                                    intIn(1) = element number
c                                    intIn(2) = element integration point number
c                                    intIn(3) = material reference number
c                                    intIn(4) = element type ID number (absolute value)
c                                               > 0 for CONTA171-CONTA177
c                                               < 0 for CONTA178
c                                    intIn(5) = real constant ID number
c                                    intIn(6) = associated contact nodal number
c                                    intIn(7) = contact indicator
c                                               0: intersection is found
c                                               otherwise: no intersection
c                                    intIn(8) = target element number
c                                    intIn(9) = flag for forcing sliding
c                                               frictional case
c                                               0 : not forcing
c                                               1 : forcing (Slip direction is
c                                                   defined through CMROT command)
c                                    intIn(10) = 1 first pass through
c                                                (1st iteration)
c                                                (useful for initializing state
c                                                variables to a non-zero value)
c                                              = 2 first pass through key of
c                                                a restart
c                                              = 3 first pass through key of
c                                                a rezoning
c                                    intIn(11) = current load step number
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                     intIn(12) = current substep number
c                                     intIn(13) = current equilibrium iteration
c                                                 number
c                                     intIn(14) = flag for using unsymmetric
c                                                 matrices (nropt,unsym)
c                                                 0 : symmetric
c                                                 1 : unsymmetric
c                                     intIn(15) = Linear perturbation flag
c                                                 0 : a general load step
c                                                 1 : a linear perturbation step
c                                     intIn(16) = key to indicate output pass
c                                                 0 : not a output pass
c                                                 1 : output pass
c                                     intIn(17) = key to indicate if history-
c                                                 dependent variables
c                                                 (user defined) need to be
c                                                 updated after the substep has
c                                                 converged
c                                                 1 : update (converged)
c                                                 0 : do not update (not converged)
c                                     intIn(18) = key to indicate transient effects
c                                                 1 : transient is active
c                                                 0 : transient is not active
c                                     intIn(19) = large deformation key [nlgeom cmd]
c                                                 1 : on
c                                                 0 : off
c                                     intIn(20) = analysis type (derived from
c                                                 antype cmd)
c                                                 0 : a static analysis
c                                                 1 : a buckling analysis
c                                                 2 : a modal analysis
c                                                 3 : a harmonic analysis
c                                                 4 : a transient analysis
c                                                 7 : a substructure analysis
c                                                 8 : a spectrum analysis
c                                     intIn(21) = key for displacement & force
c                                                 convergence
c                                                 1 : converged
c                                                 0 : not converged
c     realIn   (dp,ar(*),in)       - real variables passed in
c                                     realIn(1) = contact element length
c                                     realIn(2) = contact element depth
c                                     realIn(3) = area associated with the contact
c                                                 detection point
c                                     realIn(4) = pinball radius
c                                     realIn(5) = un-scaled normal penalty stiffness
c                                     realIn(6) = time (or frequency for a harmonic
c                                                 analysis) at the beginning of this
c                                                 load step
c                                     realIn(7) = time (or frequency for a harmonic
c                                                 analysis) at the end of this load step
c                                     realIn(8) = current time value (or frequency value
c                                                 for a harmonic analysis)
c                                     realIn(9) = time increment (or frequency increment
c                                                 for a harmonic analysis) over this
c                                                 substep
c                                     realIn(10)= temperature offset from absolute
c                                                  zero
c                                     realIn(11)= geometric penetration/gap
c                                                 (current substep)
c                                                 > 0 : gap
c                                                 < 0 : penetration
c                                     realIn(12)= time increment scaling factor to
c                                                 be used for structural transient
c                                                 dynamics
c     nkeyopt  (int,sc,in)         - number of key options
c     keyopt   (int,ar(nkeyopt),in)- array containing key options
c                                     keyopt(1) : Select degree of freedom
c                                     keyopt(2) : Contact algorithm
c                                     ... so on (see ANSYS documentation)
c     nrl      (int,sc,in)         - number of real constants
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c     rlconst  (dp,ar(nrl),in)     - array containing real constants
c                                    Elements CONTA171 to CONTA177
c                                    rlconst(1) : R1
c                                    rlconst(2) : R2
c                                    rlconst(3) : FKN
c                                    rlconst(4) : FTOLN
c                                    ... so on (see ANSYS documentation)
c                                    Element CONTA178
c                                    rlconst(1) : FKN
c                                    rlconst(2) : GAP
c                                    ... so on (see ANSYS documentation)
c
c     kcnprop  (int,sc,in)         - the position of constant in the real set
c
c                                    (see ANSYS contact element manual)
c     ncomp    (int,sc,in)         - number of stress/force component
c                                    = 9 for CONTA171-CONTA177
c                                    = 7 for CONTA178

c     stress   (dp,ar(ncomp),in)   - stress components at the beginning of
c                                    the current iteaation/substep.
c                                    stress(1) = frictional stress in direction 1
c                                    stress(2) = frictional stress in direction 2
c                                                (3D only)
c                                    stress(3) = contact normal pressure
c                                    > 0 : compression
c                                    < 0 : tension
c                                    the above contact traction must be defined in
c                                    a local coordinate system (see localr)
c                                    stress(4) = heat flux (per area)
c                                                flowing into contact
c                                    stress(5) = heat flux (per area)
c                                                flowing into target
c                                    < 0 heat flowing into a surface
c                                    > 0 heat flowing out of a surface
c                                    stress(6) = electrical current density
c                                                (or pore fluid flux density)
c                                                (per area) flowing into contact
c                                    stress(7) = electrical current density
c                                                (or pore fluid flux density)
c                                                (per area) flowing into target
c                                    > 0 current flowing out of a surface
c                                    < 0 current flowing into a surface
c                                    stress(8) = diffusion flux density
c                                                (per area) flowing into contact
c                                    stress(9) = diffusion flux density
c                                                (per area) flowing into target
c                                    > 0 flux flowing out of a surface
c                                    < 0 flux flowing into a surface
c     strain0  (dp,ar(ncomp),in) - strain components in the end of the previous
c                                    substep
c                                    (see strain for each component definition)
c     strain  (dp,ar(ncomp),in)  - current strain components
c                                    strain(1) = slip increment in direction 1
c                                    strain(2) = slip increment in direction 2
c                                              (3D only)
c                                    strain(3) = contact normal gap/penetration
c                                    < 0 : gap
c                                    > 0 : penetration
c                                    strain(4) = temperature at the contact point
c                                                (from TEMP DOF or temperature load)
c                                    strain(5) = temperature at the target point
c                                                (only from TEMP DOF)
c                                    strain(6) = voltage (or pore pressure)
c                                                at the contact point
c                                    strain(7) = voltage (or pore pressure)
c                                                at the target point
c                                    strain(8) = concentrationat the contact point
c                                    strain(9) = concentrationat the target point
c     kstat    (int,sc,in)         - contact status at the end of the previous
c                                    substep
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                     3 : stick
c                                     2 : sliding
c                                     1 : open contact (near)
c                                     0 : open contact (far)
c     mu       (dp,sc,in)           - The frictional coef at the end of previous
c                                     substep
c     coor     (dp,ar(6),in)        - Coordinates of the contact detection point
c                                     coor(1) current x
c                                     coor(2) current y
c                                     coor(3) current z
c                                     coor(4) initial x
c                                     coor(5) initial y
c                                     coor(6) initial z
c     localr   (dp,ar(3,3),in)      - the direction cosines of the local surface
c                                     coordinate system at contact detection
c                                     localr(1,1),localr(1,2),localr(1,3) in slip
c                                                                         direction 1
c                                     localr(2,1),localr(2,2),localr(2,3) in slip
c                                                                         direction 2
c                                     localr(3,1),localr(3,2),localr(3,3) in normal
c                                                                         direction
c
c     usvr     (dp,ar(nuval,nintp),inout)- additional state variables from
c                                     previous equilibrium iteration (saved
c                                     if the nsvr command is used)
c     kupdhis  (int,sc,in)          - key to indicate if history-dependent
c                                     variables (user defined) need to be
c                                     updated after the substep has converged
c                                     1 : update (converged)
c                                     0 : do not update (not converged)
c
c  output arguments:
c     variable (type,sze,intent)    description

c     cnprop   (dp,ar(5),out)       - user defined real constant value and
c                                     derivatives w.r.t. kcnprop position
c                                     cnprop(1) = user defined real constant value
c                                     (e.g. kcnprop = 3 for normal contact
c                                       stiffness FKN.
c                                       positive as scaling factor;
c                                       negative value as the absolute value)
c                                     cnprop(2) = derivative of the real constant
c                                                 w.r.t. geometric penetration/gap
c                                     cnprop(3) = derivative of the real constant
c                                                 w.r.t. contact normal pressure
c                                     cnprop(4) = derivative of the real constant
c                                                 w.r.t. temperature at contact
c                                     cnprop(5) = derivative of the real constant
c                                                 w.r.t. temperature at target
c     usvr     (dp,ar(nuval,nintp),inout)- updated additional state variables
c                                     They are passed in as the values at the
c                                     beginning of this substep.
c                                     They are updated to be the values at the
c                                     end of this substep
c                                     Use NSVR command to size usvr array and
c                                     set nuval to same value as number of
c                                     variables on NSVR commands
c                                     Use userou.F to save these values
c                                     on NMISC record for output purposes.
c                                     The number of user defined output items on
c                                     NMISC should be equal or less than NSTV
c                                     on nsvr command). It cannot exceed 120.
c
c      keyerr  (int,sc,inout)       - key to indicate if there is any element
c                                     formulation error.
c                                     The error could be caused by too
c                                     large incremental step, illegal model.
c                                     = 0 no error (present value before calling)
c                                     = 1 some error happens. ANSYS will
c                                     decide to stop the analysis or cutback
c                                     the substep (bi-section) based on other
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                   user input and information at higher
c                                   level.
c
```

2.5.2. Subroutine userfric (Writing Your Own Friction Laws)

This subroutine applies to the CONTA17*x* contact elements.

```
*deck,userfric                      USERDISTRIB
      subroutine userfric (elem,mat,intpt,nkeyopt,keyopt,nrl,rlconst,
     x ncomp,npropu,uprop,kfirst,kfsteq,kn,kt,elen,kstat,timval,
     x timinc,tcont,ttarg,toffst,dslip,slip,pres,tau,dt,usvr,
     x fdiss,elener,kupdhis,mu,dtdp)
c
c *** primary function:   Allow users to write their own friction laws.
c                         This logic is accessed with tb,fric with tbopt=user.
c                         The below demonstration logic is the same as using
c                         tb,fric for isotropic Coulomb friction.
c                         Other friction laws may require more general
c                         definition of friction forces.
c *** secondary function: demonstrate the use of user-written friction laws
c                         in this routine:
c                         a. update history variables
c                         b. compute consistent tangent matrix
c
c *** Notice - This file contains ANSYS Confidential information ***
c
c
c         Copyright ANSYS.  All Rights Reserved.
c         *** ansys, inc.
c
c  input arguments:
c     variable (type,sze,intent)    description
c
c     elem     (int,sc,in)         - element number (label)
c     mat      (int,sc,in)         - material reference number
c     intpt    (int,sc,in)         - element integration point number
c     nkeyopt  (int,sc,in)         - number of key options
c     keyopt   (int,ar(nkeyopt),in)- array containing key options
c                                     keyopt(1) : Select degree of freedom
c                                     keyopt(2) : Contact algorithm
c                                     ... so on (see ANSYS documentation)
c     nrl      (int,sc,in)         - number of real constants
c     rlconst  (dp,ar(nrl),in)     - array containing real constants
c                                     Elements CONTA171 to CONTA177
c                                     rlconst(1) : R1
c                                     rlconst(2) : R2
c                                     rlconst(3) : FKN
c                                     rlconst(4) : FTOLN
c                                     ... so on (see ANSYS documentation)
c                                     Element CONTA178
c                                     rlconst(1) : FKN
c                                     rlconst(2) : GAP
c                                     ... so on (see ANSYS documentation)
c     ncomp    (int,sc,in)         - no. of friction stress components (1 or 2)
c     npropu   (int,sc,in)         - no. of user-defined friction properties
c     uprop    (dp,ar(npropu),in)  - user-defined material properties
c     kfirst   (int,sc,in)         - 1 if first time through, 0 otherwise
c                                     (useful for initializing state variables
c                                     to a non-zero value)
c     kfsteq   (int,sc,in)         - 1 if first equilibrium iteration of a
c                                     substep, 0 otherwise
c     kn       (dp,sc,in)          - normal penalty stiffness
c     kt       (dp,sc,in)          - tangential penalty stiffness
c                                     (an initial guess is provided but
c                                      the user must pick a suitable
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                     value that allows minimal tangential
c                                     slip during sticking without
c                                     adversely affecting the convergence;
c                                     a possible choice could be kt=mu*kn).
c                                     For Lagrange multiplier method (keyopt(2)=4
c                                     use a small kt (several orders of magnitude
c                                     smaller than mu*pres).
c     elen     (dp,sc,in)           - length of contact element
c     kstat    (int,sc,inout)       - contact status
c                                     3 : stick
c                                     2 : sliding
c                                     1 : open contact (near)
c                                     0 : open contact (far)
c     timval   (dp,sc,in)           - current time value
c     timinc   (dp,sc,in)           - time increment over this substep
c     tcont    (dp,sc,in)           - contact surface temperature
c                                     (from temperature DOF or temperature load)
c     ttarg    (dp,sc,in)           - target surface temperature
c                                     (only from temperature DOF)
c     toffst   (dp,sc,in)           - temperature offset from absolute zero
c     dslip    (dp,ar(ncomp),in)    - slip increment (current substep)
c     slip     (dp,ar(ncomp),inout)- accumulated slip (previous substep)
c     pres     (dp,sc,in)           - normal pressure/force (current substep)
c                                     > 0 : compression
c                                     < 0 : tension
c     tau      (dp,ar(ncomp),inout)- frictional stress (previous substep)
c                                     Lagrange multiplier contribution is added
c                                     if keyopt(2)=4
c     usvr     (dp,ar(nuval,nintp),inout)- additional state variables from
c                                     previous equilibrium iteration (saved
c                                     if the nsvr command is used)
c     kupdhis  (int,sc,in)          - key to indicate if history-dependent
c                                     variables (user defined) need to be
c                                     updated after the substep has converged
c                                     1 : update (converged)
c                                     0 : do not update (not converged)
c
c  output arguments:
c     variable (type,sze,intent)    description
c
c     kstat    (int,sc,inout)       - updated contact status
c     mu       (dp,sc  inout)       - updated friction coefficient
c     slip     (dp,ar(ncomp),inout)- updated accumulated slip
c     tau      (dp,ar(ncomp),inout)- updated frictional stress
c     dt       (dp,ar(5,5),out)     - material tangent modulus
c                                     rows and columns of dt matrix are
c                                     associated to:
c                                     row 1 : frictional stress in direction 1
c                                     row 2 : frictional stress in direction 2
c                                     row 3 : normal pressure
c                                     row 4 : blank
c                                     row 5 : blank
c                                     col 1 : sliding in direction 1
c                                     col 2 : sliding in direction 2
c                                     col 3 : normal gap
c                                     col 4 : temperature at contact
c                                     col 5 : temperature at targte
c                                     relevant components to be filled in are:
c                                     dt(1,1): d(tau1)/d(slip1)
c                                     dt(1,2): d(tau1)/d(slip2)
c                                     dt(1,3): d(tau1)/d(normal gap)
c                                     dt(2,1): d(tau2)/d(slip1)
c                                     dt(2,2): d(tau2)/d(slip2)
c                                     dt(2,3): d(tau2)/d(normal gap)
c                                     dt(3,3): d(pres)/d(normal gap)
c                                     dt(3,3) set to kn internally
c                                     dt(1,4) : d(tau1)/d(tcont)
c                                     dt(1,5) : d(tau1)/d(ttarg)
c                                     dt(2,4) : d(tau2)/d(tcont)
c                                     dt(2,5) : d(tau2)/d(ttarg)
c     dtdp     (dp,ar(ncomp),out)   - partial derivative of the frictional
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                   stress in direction 1/2 w.r.t. normal
c                                   pressure used in Lagrange multiplier
c                                   method (keyopt(2)=3,4).
c     usvr     (dp,ar(nuval,nintp),inout)- updated additional state variables
c                                   For example, mu value and absolute
c                                   accumulated slip could be output as follows:
c                                   usvr(1,intpt) : mu
c                                   usvr(2,intpt) : abs. acc. slip in dir1
c                                   usvr(3,intpt) : abs. acc. slip in dir2
c                                   Use NSVR command to size usvr array and
c                                   set nuval to same value as number of
c                                   variables on NSVR commands
c                                   Use userou.F to save these values
c                                   on NMISC record for output purposes.
c                                   The number of user defined output items on
c                                   NMISC should be equal or less than NSTV
c                                   on nsvr command). It cannot exceed 120.
c
c     fdiss    (dp,sc,out)         - incremental frictional dissipation
c                                     per unit area
c     elener   (dp,sc,out)         - incremental elastic stored energy
c                                     per unit area
c
c  fortran parameters (to be defined by the user):
c     variable (type)              description
c     nuval    (int)               - number of additional state variables per
c                                     integration point
c     nintp    (int)               - maximum number of integration points of
c                                     an element to be used with this routine
c                                     (14 is the maximum)
c         note:  nuval x nintp = nstv(on nsvr command); cannot exceed 840!
c
c  internal variables:
c     variable (type,sze)      description
c     dtfac    (dp,sc)         - temporary variable
c     taulim   (dp,sc)         - limit frictional stress
c     taueq    (dp,sc)         - equivalent frictional stress
c     dir1     (dp,sc)         - slip increment direction 1
c     dir2     (dp,sc)         - slip increment direction 2
c     dslipeq  (dp,sc)         - equivalent slip increment
c     oldt1    (dp,sc)         - frictional stress 1 from prev substep
c     oldt2    (dp,sc)         - frictional stress 2 from prev substep
c     err      (dp,ar(2))      - data array for diagnostic message
c
```

2.5.3. Subroutine userinter (Writing Your Own Contact Interactions)

This subroutine applies to the CONTA17*x* contact elements.

```
*deck,userinter                       USERDISTRIB
      subroutine userinter (ndim,coor,nkeyopt,keyopt,nrl,rlconst,
     x  npropu,uprop,nintIn,intIn,nrealIn,realIn,kupdhis,localr,
     x  nuval,nintp,usvr,ncomp,stress,strain0,strain,
     x  kstat,mu,dt,dtdp,kdamp,damp,fdiss,elener,keyerr,keycnv)
c
c *** primary function:   Allow users to write their own interaction behavior.
c                         This logic is accessed with tb,inter with tbopt=user.
c *** secondary function: demonstrate the use of user-written interface laws
c                         in this routine:
c                         a. update history variables
c                         b. compute consistent tangent matrix
c
c *** Notice - This file contains ANSYS Confidential information ***
c
c
c         *** Copyright ANSYS.  All Rights Reserved.
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c         *** ansys, inc.
c
c  input arguments:
c     variable (type,sze,intent)    description
c
c     elem     (int,sc,in)         - element number
c     intpt    (int,sc,in)         - element integration point number
c     ndim     (int,sc,in)         - number of dimensions of the problem
c                                    = 2 2D
c                                    = 3 3D
c     nintp    (int,sc,in)         - the total number of integration points of
c                                    an element to be used with this routine
c     nuval    (int)               - number of additional state variables per
c                                    integration point
c         note:  nuval x nintp = nstv(on nsvr command); cannot exceed 840!
c
c     intIn    (int,ar(*),in)      - integer variables passed in
c                                    intIn(1) = element number
c                                    intIn(2) = element integration point number
c                                    intIn(3) = material reference number
c                                    intIn(4) = element type ID number (absolute value)
c                                               > 0 for CONTA171-CONTA177
c                                               < 0 for CONTA178
c                                    intIn(5) = real constant ID number
c                                    intIn(6) = associated contact nodal number
c                                    intIn(7) = contact indicator
c                                               0: intersection is found
c                                               otherwise: no intersection
c                                    intIn(8) = target element number
c                                    intIn(9) = flag for forcing sliding
c                                               frictional case
c                                               0: - not forcing
c                                               1: - forcing (Slip direction is
c                                                    defined through CMROT command)
c                                    intIn(10) = 1 first pass through
c                                                  (1st iteration)
c                                                  (useful for initializing state
c                                                   variables to a non-zero value)
c                                              = 2 first pass through key of
c                                                  a restart
c                                              = 3 first pass through key of
c                                                  a rezoning
c                                    intIn(11) = current load step number
c                                    intIn(12) = current substep number
c                                    intIn(13) = current equilibrium iteration number
c                                    intIn(14) = flag for using unsymmetric matrices
c                                                (nropt,unsym)
c                                                0: - symmetric
c                                                1: - unsymmetric
c                                    intIn(15) = Linear perturbation flag
c                                                0: - a general load step
c                                                1: - a linear perturbation step
c                                    intIn(16) = key to indicate output pass
c                                                0: not a output pass
c                                                1: output pass
c                                    intIn(17) = key for displacement & force
c                                                convergence
c                                                1: converged
c                                                0: not converged
c                                    intIn(18) = key to indicate transient effects
c                                                1 : transient is active
c                                                0 : transient is not active
c                                    intIn(19) = large deformation key [nlgeom cmd]
c                                                1 : on
c                                                0 : off
c                                    intIn(20) = analysis type (derived from antype)
c                                                0 : a static analysis
c                                                1 : a buckling analysis
c                                                2 : a modal analysis
c                                                3 : a harmonic analysis
c                                                4 : a transient analysis
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                              7 : a substructure analysis
c                                              8 : a spectrum analysis
c     realIn   (dp,ar(*),in)       - real variables passed in
c                                    realIn(1) = contact element length
c                                    realIn(2) = contact element depth
c                                    realIn(3) = area associated with the contact
c                                                detection point
c                                    realIn(4) = pinball radius
c                                    realIn(5) = unscaled normal penalty stiffness
c                                    realIn(6) = time (or frequency for a harmonic
c                                                analysis) at the beginning of this
c                                                load step
c                                    realIn(7) = time (or frequency for a harmonic
c                                                analysis) at the end of this load step
c                                    realIn(8) = current time value (or frequency value
c                                                for a harmonic analysis)
c                                    realIn(9) = time increment (or frequency increment
c                                                for a harmonic analysis) over this
c                                                substep
c                                    realIn(10) = temperature offset from absolute
c                                                 zero
c                                    realIn(11) = geometric penetration/gap
c                                                 (current substep)
c                                                 > 0 : gap
c                                                 < 0 : penetration
c                                    realIn(12) = time increment scaling factor to
c                                                 be used for structural transient
c                                                 dynamics
c                                    realIn(13) = convection coefficient (SFE command)
c                                    realIn(14) = bulk temp (SFE command)
c     nkeyopt  (int,sc,in)         - number of key options
c     keyopt   (int,ar(nkeyopt),in)- array containing key options
c                                    keyopt(1) : Select degree of freedom
c                                    keyopt(2) : Contact algorithm
c                                    ... so on (see ANSYS documentation)
c     nrl      (int,sc,in)         - number of real constants
c     rlconst  (dp,ar(nrl),in)     - array containing real constants
c                                    Elements CONTA171 to CONTA177
c                                    rlconst(1) : R1
c                                    rlconst(2) : R2
c                                    rlconst(3) : FKN
c                                    rlconst(4) : FTOLN
c                                    ... so on (see ANSYS documentation)
c                                    Element CONTA178
c                                    rlconst(1) : FKN
c                                    rlconst(2) : GAP
c                                    ... so on (see ANSYS documentation)
c     ncomp    (int,sc,in)         - number of stress/force component
c                                    = 9 for CONTA171-CONTA177
c                                    = 7 for CONTA178
c     stress   (dp,ar(ncomp),inout)- stress components (current substep)
c                                    It is passed in as the stress at the beginning
c                                    of the current substep. It is updated to be
c                                    the stress at the end of this current substep
c                                    stress(1) = frictional stress in direction 1
c                                    stress(2) = frictional stress in direction 2
c                                                (3D only)
c                                    stress(3) = contact normal pressure
c                                    > 0 : compression
c                                    < 0 : tension
c                                    the above contact traction must be defined in
c                                    a local coordinate system (see localr)
c                                    Lagrange multiplier contribution is added
c                                    if keyopt(2)=3,4
c                                    stress(4) = heat flux (per area)
c                                                flowing into contact
c                                    stress(5) = heat flux (per area)
c                                                flowing into target
c                                    < 0 heat flowing into a surface
c                                    > 0 heat flowing out of a surface
c                                    stress(6) = electrical current density
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                                (or pore fluid flux density)
c                                                (per area) flowing into contact
c                                     stress(7) = electrical current density
c                                                (or pore fluid flux density)
c                                                (per area) flowing into target
c                                     > 0 current flowing out of a surface
c                                     < 0 current flowing into a surface
c                                     stress(8) = diffusion flux density
c                                                (per area) flowing into contact
c                                     stress(9) = diffusion flux density
c                                                (per area) flowing into target
c                                     > 0 flux flowing out of a surface
c                                     < 0 flux flowing into a surface
c     strain0  (dp,ar(ncomp),in)    - strain components in the end of the previous
c                                     substep
c                                     (see strain for each component definition)
c     strain   (dp,ar(ncomp),in)    - current strain components
c                                     strain(1) = slip increment in direction 1
c                                     strain(2) = slip increment in direction 2
c                                                 (3D only)
c                                     strain(3) = contact normal gap/penetration
c                                     < 0 : gap
c                                     > 0 : penetration
c                                     strain(4) = temperature at the contact point
c                                                (from TEMP DOF or temperature load)
c                                     strain(5) = temperature at the target point
c                                                (only from TEMP DOF)
c                                     strain(6) = voltage (or pore pressure)
c                                                 at the contact point
c                                     strain(7) = voltage (or pore pressure)
c                                                 at the target point
c                                     strain(8) = concentrationat the contact point
c                                     strain(9) = concentrationat the target point
c
c     kstat    (int,sc,inout)       - contact status (current substep)
c                                     It is passed in as the status at the
c                                     beginning of the current substep.
c                                     It is updated to be the status at the
c                                     end of the current substep
c                                     3 : stick
c                                     2 : sliding
c                                     1 : open contact (near)
c                                     0 : open contact (far)
c     coor     (dp,ar(6),in)        - Coordinates of the contact detection point
c                                     coor(1) current x
c                                     coor(2) current y
c                                     coor(3) current z
c                                     coor(4) initial x
c                                     coor(5) initial y
c                                     coor(6) initial z
c     localr   (dp,ar(3,3),in)      - the direction cosines of the local surface
c                                     coordinate system at contact detection
c                                     localr(1,1),localr(1,2),localr(1,3) in
c                                                                slip direction 1
c                                     localr(2,1),localr(2,2),localr(2,3) in
c                                                                slip direction 2
c                                     localr(3,1),localr(3,2),localr(3,3) in
c                                                                 normal direction
c
c     npropu   (int,sc,in)          - number of user-defined interaction properties
c     uprop    (dp,ar(npropu),in)   - user-defined material properties
c
c     usvr     (dp,ar(nuval,nintp),inout)- additional state variables from
c                                       previous equilibrium iteration (saved
c                                       if the nsvr command is used)
c     kupdhis  (int,sc,in)          - key to indicate if history-dependent
c                                     variables (user defined) need to be
c                                     updated after the substep has converged
c                                     1 : update (converged)
c                                     0 : do not update (not converged)
c
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c  output arguments:
c     variable (type,sze,intent)   description
c
c     kstat    (int,sc,inout)      - updated contact status
c     stress   (dp,ar(ncomp),inout)- updated stress components
c
c     dt   (dp,ar(ncomp,ncomp),out)- interface stiffness matrix:
c                                    dt(i,j) defines the partial derivative of
c                                    the ith stress component at the current
c                                    substep w.r.t. the jth component of the
c                                    relative strain increment array.
c                                    If symmetric solver option used, ANSYS will
c                                    symmetrize the matrix bu averaging the
c                                    off-diagonal terms.
c                                    rows and columns of dt matrix are
c                                    associated to:
c                                    row 1 : frictional stress in direction 1
c                                    row 2 : frictional stress in direction 2
c                                    row 3 : normal pressure
c                                       > 0 : compression
c                                       < 0 : tension
c                                    row 4 : heat flux out the contact surface
c                                       < 0 heat flowing into contact
c                                       > 0 heat flowing out of target
c                                    row 5 : heat flux out the target surface
c                                       < 0 heat flowing into target
c                                       > 0 heat flowing out of target
c                                    row 6 : electrical current density
c                                            (or pore prssure)
c                                            flowing out the contact surface
c                                       > 0 current flowing out of contact
c                                       < 0 current flowing into contact
c                                    row 7 : electrical current density
c                                            (or pore prssure)
c                                       > 0 current flowing out of target
c                                       < 0 current flowing into target
c                                    row 8 : diffusion flux density
c                                            flowing out the contact surface
c                                       > 0 flux flowing out of contact
c                                       < 0 flux flowing into contact
c                                    row 9 : diffusion flux density
c                                       > 0 flux flowing out of target
c                                       < 0 flux flowing into target
c                                    col 1 : sliding in direction 1
c                                    col 2 : sliding in direction 2
c                                    col 3 : normal gap
c                                       < 0 : gap
c                                       > 0 : penetration
c                                    col 4 : temperature at the contact surface
c                                    col 5 : temperature at the target surface
c                                    col 6 : voltage at the contact surface
c                                    col 7 : voltage at the target surface
c                                    col 8 : concentration at the contact surface
c                                    col 9 : concentration at the target surface
c                                    relevant components to be filled in are:
c                                    dt(1,1): d(tau1)/d(slip1)
c                                    dt(1,2): d(tau1)/d(slip2)
c                                    dt(1,3): d(tau1)/d(normal gap)
c                                    dt(1,4): d(tau1)/d(tempC)
c                                    dt(1,5): d(tau1)/d(tempT)
c                                    dt(1,6): d(tau1)/d(voltC)
c                                    dt(1,7): d(tau1)/d(voltT)
c                                    dt(1,8): d(tau1)/d(concC)
c                                    dt(1,9): d(tau1)/d(concT)
c                                    dt(2,1): d(tau2)/d(slip1)
c                                    dt(2,2): d(tau2)/d(slip2)
c                                    dt(2,3): d(tau2)/d(normal gap)
c                                    ...
c                                    dt(3,1): d(pres)/d(slip 1)
c                                    dt(3,2): d(pres)/d(slip 2)
c                                    dt(3,3): d(pres)/d(normal gap)
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                     ...
c                                     dt(4,1): d(fluxC)/d(slip 1)
c                                     dt(4,2): d(fluxC)/d(slip 2)
c                                     dt(4,3): d(fluxC)/d(normal gap)
c                                     dt(4,4): d(fluxC)/d(tempC)
c                                     dt(4,5): d(fluxC)/d(tempT)
c                                     dt(4,6): d(fluxC)/d(voltC)
c                                     dt(4,7): d(fluxC)/d(voltT)
c                                     dt(4,8): d(fluxC)/d(concC)
c                                     dt(4,9): d(fluxC)/d(concT)
c                                     ...
c                                     dt(5,4): d(fluxT)/d(tempC)
c                                     dt(5,5): d(fluxT)/d(tempT)
c                                     dt(5,6): d(fluxT)/d(voltC)
c                                     dt(5,7): d(fluxT)/d(voltT)
c                                     dt(5,8): d(fluxT)/d(concC)
c                                     dt(5,9): d(fluxT)/d(concT)
c                                     ...
c                                     dt(6,4): d(eleC)/d(tempC)
c                                     dt(6,5): d(eleC)/d(tempT)
c                                     dt(6,6): d(eleC)/d(voltC)
c                                     dt(6,7): d(eleC)/d(voltT)
c                                     dt(6,8): d(eleC)/d(concC)
c                                     dt(6,9): d(eleC)/d(concT)
c                                     ...
c                                     dt(7,4): d(eleT)/d(tempC)
c                                     dt(7,5): d(eleT)/d(tempT)
c                                     dt(7,6): d(eleT)/d(voltC)
c                                     dt(7,7): d(eleT)/d(voltT)
c                                     dt(7,8): d(eleT)/d(concC)
c                                     dt(7,9): d(eleT)/d(concT)
c                                     ...
c                                     dt(8,4): d(diffC)/d(tempC)
c                                     dt(8,5): d(diffC)/d(tempT)
c                                     dt(8,6): d(diffC)/d(voltC)
c                                     dt(8,7): d(diffC)/d(voltT)
c                                     dt(8,8): d(diffC)/d(concC)
c                                     dt(8,9): d(diffC)/d(concT)
c                                     ...
c                                     dt(9,4): d(diffT)/d(tempC)
c                                     dt(9,5): d(diffT)/d(tempT)
c                                     dt(9,6): d(diffT)/d(voltC)
c                                     dt(9,7): d(diffT)/d(voltT)
c                                     dt(9,8): d(diffT)/d(concC)
c                                     dt(9,9): d(diffT)/d(concT)
c     dtdp     (dp,ar(ncomp),out)  - partial derivative of the frictional stress
c                                     in direction 1,2 w.r.t. normal pressure
c                                     used in Lagrange multiplier method
c                                     (keyopt(2)=3,4).
c     damp     (dp,ar(3,3),out)    - interface damping matrix (structure only)
c                                     it can be used only in Linear perturbation
c                                     modal analysis or transient analysis or
c                                     harmonic analysis in frequence domain.
c                                     damp(i,j) defines the partial derivative of
c                                     the ith stress component at the current
c                                     substep w.r.t. the jth component of the
c                                     strain increment rate array.
c                                     rows and columns of dt matrix are
c                                     associated to:
c                                     row 1 : frictional stress in direction 1
c                                     row 2 : frictional stress in direction 2
c                                     row 3 : normal pressure
c                                     col 1 : sliding rate in direction 1
c                                     col 2 : sliding rate in direction 2
c                                     col 3 : normal gap rate
c     kdamp    (int,sr,out)        - damping matrix index
c                                     0 : no damping matrix
c                                     1 : taking damping matrix into account
c     usvr     (dp,ar(nuval,nintp),inout)- updated additional state variables
c                                     For example, mu value and absolute/relative
c                                     accumulated slip could be output as follows:
```

```
c                                     usvr(1,intpt) : mu
c                                     usvr(2,intpt) : abs. acc. slip in dir1
c                                     usvr(3,intpt) : abs. acc. slip in dir2
c                                     usvr(4,intpt) : acc. slip in dir1
c                                     usvr(5,intpt) : acc. slip in dir2
c                                     They are passed in as the values at the
c                                     beginning of this substep. They are updated
c                                     to be the values at the end of this substep.
c                                     Use NSVR command to size usvr array and
c                                     set nuval to same value as number of
c                                     variables on NSVR commands
c                                     Use userou.F to save these values
c                                     on NMISC record for output purposes.
c                                     The number of user defined output items on
c                                     NMISC should be equal or less than NSTV
c                                     on nsvr command). It cannot exceed 120.
c
c     mu       (dp,sc,inout)       - The current frictional coefficient
c     fdiss    (dp,sc,out)         - incremental frictional dissipation
c                                     per unit area
c     elener   (dp,sc,inout)       - Total elastic stored energy
c                                     per unit area.
c                                     Previous converged value is passed in and
c                                     current total should be the output
c
c     keyerr (int,sc,out)          - key to indicate if there is any element
c                                     formulation error, like
c                                     contact status changes abruptly,
c                                     too much penetration.
c                                     The error could be caused by too
c                                     large incremental step, illegal model.
c                                     = 0 no error (preset value before calling)
c                                     = 1 some error happens. ANSYS will
c                                     decide to stop the analysis or cutback
c                                     the substep (bi-section) based on other
c                                     user input and information at higher
c                                     level.
c     keycnv (int,sc,inout)        - key to flag if this element satisfies
c                                     the user defined element convergence
c                                     criterion.
c                                     = 1, yes, the criterion is satisfied
c                                       or don't have any criterion at all
c                                       it is preset value before calling
c                                     = 0, no, the element doesn't satisfy
c                                       element convergence criterion. If
c                                       this is the case, the iteration will
c                                       not converge even when both force
c                                       and displacement converge
c
c  internal variables:
c     variable (type,sze)      description
c
```

2.5.4. Subroutine userwear (Writing Your Own Wear Law)

This subroutine applies to CONTA17*x* contact elements.

```
*deck,userwear                        USERDISTRIB
      subroutine userwear(WearInc,WearDir,TotWearOld,strain,
     x     stress,temperature,dtime,YieldStress,nTbprop,Tbprop,
     x     coor,kstat,elem,intpt,ndim,localr,intIn,realIn,usvr,
     x     keyopt,rlconst)
c *** Primary Function:   Calculates the Wear Increment and (optionally) wear direction
c *** Notice - This file contains ANSYS Confidential information ***
c
c
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c         *** Copyright ANSYS.  All Rights Reserved.
c         *** ansys, inc.
c input arguments:
c     variable (type,sze,intent)    description
c
c     ndim     (int,sc,in)         - number of dimensions of the problem
c                                    = 2 2D
c                                    = 3 3D
c     TotWearOld(dp,ar(ndim),in)   - Total Wear at the contact point at the previous substep
c     strain   (dp,ar(3),in)   - current strain components in contact surface coordinate system
c                                    strain(1) = slip increment in direction 1
c                                    strain(2) = slip increment in direction 2
c                                                (3D only)
c                                    strain(3) = contact normal gap/penetration
c                                    < 0 : gap
c                                    > 0 : penetration
c     stress   (dp,ar(3),in)   - stress components  in contact surface coordinate system
c                                    stress(1) = frictional stress in direction 1
c                                    stress(2) = frictional stress in direction 2
c                                                (3D only)
c                                    stress(3) = contact normal pressure
c                                    > 0 : compression
c                                    < 0 : tension
c     temperature  (dp,sc,in)      - temperature
c     dtime (dp,sc,in)             - time increment
c     YieldStress (dp,sc,in)       - Yield stress of underlying element (defined only for Plastic material-see do
c     nTbprop (int,sc,in)          - Number of TBdata for Tb,Wear per field
c     Tbprop (dp,ar(nTbprop,in)    - TB data for the the Tb,Wear option at the given temperature
c     coor     (dp,ar(6),in)       - Coordinates of the contact detection point
c                                     coor(1) current x
c                                     coor(2) current y
c                                     coor(3) current z
c                                     coor(4) initial x
c                                     coor(5) initial y
c                                     coor(6) initial z
c     kstat    (int,sc,in)          - contact status (current substep)
c                                     3 : stick
c                                     2 : sliding
c                                     1 : open contact (near)
c                                     0 : open contact (far)
c     elem     (int,sc,in)         - element number
c     intpt    (int,sc,in)         - element integration point number
c     localr   (dp,ar(3,3),in)     - the direction cosines of the local surface
c                                    coordinate system at contact detection
c                                    localr(1,1),localr(1,2),localr(1,3) in
c                                                            slip direction 1
c                                    localr(2,1),localr(2,2),localr(2,3) in
c                                                            slip direction 2
c                                    localr(3,1),localr(3,2),localr(3,3) in
c                                                             normal direction
c     intIn    (int,ar(*),in)      - integer variables passed in
c                                     intIn(1) = target element number the contact element
c                                                is in contat with (or is closest)
c                                     intIn(2) = Attached contact element number (if any)
c                                                to the target element in passed in intIn(1)
c                                     intIn(3) = number of additional state variables per
c                                                integration point (nuval)
c                                     intIn(4) = the total number of integration points of
c                                                an element to be used with this routine (nintp)
c                                     intIn(5) = key to indicate if history-dependent
c                                                variables (user defined) need to be
c                                                updated after the substep has converged
c                                                1 : update (converged)
c                                                0 : do not update (not converged)
c     realIn   (dp,ar(*),in)       - real variables passed in
c                                    realIn(1) = contact element length
c                                    realIn(2) = contact element depth
c                                    realIn(3) = area associated with the contact
c                                                detection point
c
c     usvr     (dp,ar(nuval*nintp),inout)- additional state variables from
```

```
c                                    previous equilibrium iteration (saved
c                                    if the nsvr command is used
c     keyopt   (int,ar(*),in)     - array containing key options for the element
c     rlconst  (dp,ar(*),in)      - array containing real constants for the element

c Output Arguments:
c     variable (type,sze,intent)   description
c
c     WearInc  (dp,sc)             - Increment in the Wear (magnitude)- User must define
c     WearDir  (dp,ar(ndim),inout) - Direction cosines in global coordinate system
c                                    in which wear increment will be applied- Optional
c                                    default coming in -Contact normal direction
```

2.6. Subroutines for Customizing Loads

The following subroutines modify or monitor existing element loading:

Activate these subroutines via **USRCAL**.

2.6.1. Subroutine usrefl (Changing Scalar Fields to User-Defined Values)

```
*deck,usrefl                      USERDISTRIB
      subroutine usrefl (key,iel,ielc,nnod,nodes,time,defalt,nd,dat)
c *** primary function:  change the scalar fields (temperatures, fluences,
c                 heat generation, etc.) to what user desires.
c *** secondary functions: none
c
c         in order to activate this user programmable feature,
c         the user must enter the usrcal command.
c
c         this routine is called at each substep of each load step
c         for which element or nodal temperatures(etc) are used.
c         it is called for each equilibrium iteration.
c         the call to get the standard ansys input element or nodal values
c         is made just before entering this routine.
c
c         *** Copyright ANSYS.  All Rights Reserved.
c         *** ansys, inc.
c
c *** Notice - This file contains ANSYS Confidential information ***
c
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c     typ=int,dp,log,chr,dcp   siz=sc,ar(n)   intent=in,out,inout
c
c  input arguments:
c     variable (typ,siz,intent)    description
c     key      (int,sc,in)       - type of data desired
c                                  = 1 temperatures
c                                  = 2 fluences
c                                  = 3 heat generation rates
c                                  = 4 moisture contents
c                                  = 5 magnetic virtual displacements
c     iel      (int,sc,in)       - element number
c     ielc   (int,ar(IELCSZ),in) - array of element type characteristics
c     nnod     (int,sc,in)       - number of nodes
c     nodes    (int,ar(nnod),in) - list of nodes
c     time     (dp,sc,in)        - time of current substep
c     defalt   (dp,sc,in)        - default value (e.g. tunif)
c     nd       (int,sc,in)       - size of dat array
c     dat      (dp,ar(nd),inout) - array of data as normally computed by element
c                                  as selected by key
c
c  output arguments:
c     variable (typ,siz,intent)    description
c     dat      (dp,ar(nd),inout) - array of data passed back to element
c                                    this data represents values at the end
c                                    of the load step
c
c        the input argument dat may be used in one of three ways:
c            1.  it may be simply passed thru
c            2.  it may be used as a flag(e.g. if dat(1) = -3.0, use
c                                 a certain set of logic)
c            3.  it may be completely ignored and instead defined with new logic
c
```

2.6.2. Subroutine userpr (Changing Element Pressure Information)

```
*deck,userpr                       USERDISTRIB
      subroutine userpr (ielc,elem,time,ndat,dat)
c *** primary function:    change element pressure information.

c         *** Copyright ANSYS.  All Rights Reserved.
c         *** ansys, inc.

c         in order to activate this user programmable feature,
c         the user must enter the 'usrcal,userpr' command.

c         this routine is called at each substep of each load step for which
c         pressures are used. it is called for each equilibrium iteration.
c         it is called once per element.
c         the call to get the standard ansys input pressures is made just before
c         entering this routine.

c     input arguments:
c        variable (typ,siz,intent)    description
c        ielc   (int,ar(IELCSZ),in) - array of element type characteristics
c        elem     (int,sc,in)       - element number for operation.
c        time     (dp,sc,in)        - time of current substep
c        ndat     (int,sc,in)       - number of pressure items for this element
c        dat      (dp,ar(ndat,2),inout) - the element pressure vector
c                                      (has input values for each corner
c                                      of each face)


c     output arguments:
c        variable (typ,siz,intent)    description
c        dat      (dp,ar(ndat,2),inout) - the element pressure vector
c                                      (defines input values for each corner
c                                      of each face)
```

```
c                                     dat(1:ndat,1) - real pressures
c                                     dat(1:ndat,2) - complex pressures
c                                                     (surface elements only)


c        the input array  dat  may be used in one of three ways:
c          1.  it may be simply passed thru
c          2.  it may be used as a flag(e.g. if dat(1) = -3.0, use
c                 a certain set of logic)
c          3.  it may be completely ignored and instead defined with new logic
```

2.6.3. Subroutine usercv (Changing Element Face Convection Surface Information)

```
*deck,usercv                       USERDISTRIB
      subroutine usercv (elem,ielc,time,nr,u, ndat,hc,tb)
c *** primary function: change element face convection surface info
c
c         *** Copyright ANSYS.  All Rights Reserved.
c         *** ansys, inc.
c
c         in order to activate this user programmable feature,
c         the user must enter the 'usrcal,usercv' command.
c
c        the input arguments hc and tb may be used in one of three ways:
c           1.  they may be simply passed thru.
c           2.  they may be used as a flag(e.g. if hc(2) = -3.0, use
c                                a certain set of logic).
c           3.  they may be completely ignored.
c                                 and instead redefined with new logic

c         this routine is called during each substep of each load step.
c         it is called for each equilibrium iteration.
c         it is called once per element.  it is called only during the heat
c         flow load vector formulation stage, and not during the heat flow
c         evaluation stage.
c         the call to get the standard ansys input convection surfaces
c         is made just before entering this routine, so this information is
c         available to be modified, if desired.
c
c         velocity-dependent film coefficients can be computed by inputting the
c         velocity as the input film coefficient or bulk temperature or
c         by inputting the velocity as a function of location in space.  this
c         routine could then compute the effective film coefficient.


c
c     input arguments:
c        variable (typ,siz,intent)    description
c        elem     (int,sc,in)       - element number for operation.
c        ielc   (int,ar(IELCSZ),in) - array of element type characteristics
c        time     (dp,sc,in)        - time of current substep
c        nr       (int,sc,in)       - number of nodal temperatures
c                                         of the element
c        u        (dp,ar(nr),in)    - vector of most recent values of the
c                                     temperatures
c        ndat     (int,sc,in)       - number of data points per element
c                                     for example, for solid70, ndat = 24 = 6*4
c                                     where 6 = faces per element
c                                           4 = corners per face
c        hc     (dp,ar(ndat),inout) - film coefficients
c                                     (has input values for each corner
c                                     of each face)
c        tb     (dp,ar(ndat),inout) - bulk temperature
c                                     (has input values for each corner
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                     of each face)
c
c     output arguments:
c        variable (typ,siz,intent)    description
c        hc     (dp,ar(ndat),inout) - film coefficients
c                                      (defines input values for each corner
c                                      of each face)
c        tb     (dp,ar(ndat),inout) - bulk temperature
c                                      (defines input values for each corner
c                                      of each face)
c
```

2.6.4. Subroutine userfx (Changing Element Face Heat Flux Surface Information)

```
*deck,userfx                       USERDISTRIB
      subroutine userfx (ielc,elem,time,nr,u, ndat,dat)
c *** primary function: change element face heat flux surface info
c
c         *** Copyright ANSYS.  All Rights Reserved.
c         *** ansys, inc.
c
c         in order to activate this user programmable feature,
c         the user must enter the 'usrcal,userfx' command.
c
c         this routine is called during each substep of each load step.
c         it is called for each equilibrium iteration.
c         it is called once per element.  it is called only during the heat
c         flow load vector formulation stage, and not during the heat flow
c         evaluation stage.
c         the call to get the standard ansys input heat flux surfaces
c         is made just before entering this routine, so this information is
c         available to be modified, if desired.
c
c     input arguments:
c        variable (typ,siz,intent)    description
c        ielc   (int,ar(IELCSZ),in) - array of element type characteristics
c        elem     (int,sc,in)       - element number for operation.
c        time     (dp,sc,in)        - time of current substep
c        nr       (int,sc,in)       - number of nodal temperatures
c                                         of the element
c        u        (dp,ar(nr),in)    - vector of most recent values of the
c                                      temperatures
c        ndat     (int,sc,in)       - number of data points per element
c                                      for example, for solid70, ndat = 24 = 6*4
c                                      where 6 = faces per element
c                                            4 = corners per face
c        dat    (dp,ar(ndat),inout) - fluxes
c                                      (has input values for each corner
c                                      of each face)

c
c     output arguments:
c        variable (typ,siz,intent)    description
c        dat    (dp,ar(ndat),inout) - fluxes
c                                      (defines input values for each corner
c                                      of each face)
c
```

2.6.5. Subroutine userch (Changing Element Face Charge Density Surface Information)

```
*deck,userch                      USERDISTRIB
      subroutine userch (ielc,ielem,time,nr,u, ndat,dat)
c *** primary function: change element face charge density surface info
c
c         in order to activate this user programmable feature,
c         the user must enter the usrcal command.
c
c         this routine is called during each substep of each load step.
c         it is called once per element.  it is called only during the heat
c         flow load vector formulation stage, and not during the heat flow
c         evaluation stage.
c         the call to get the standard ansys input charge densities of surfaces
c         is made just before entering this routine, so this information is
c         available to be modified, if desired.
c
c         *** Copyright ANSYS.  All Rights Reserved.
c         *** ansys, inc.
c
c     input arguments:
c        variable (typ,siz,intent)    description
c        ielc   (int,ar(IELCSZ),in) - array of element type characteristics
c        ielem    (int,sc,in)       - element number for operation.
c        time     (dp,sc,in)        - time of current substep
c        nr       (int,sc,in)       - number of nodal temperatures
c                                         of the element
c        u        (dp,ar(nr),in)    - vector of most recent values of the
c                                      temperatures
c        ndat     (int,sc,in)       - number of data points per element
c        dat    (dp,ar(ndat),inout) - fluxes
c
c     output arguments:
c        variable (typ,siz,intent)    description
c        dat    (dp,ar(ndat),inout) - fluxes
c
c        the input argument dat may be used in one of three ways:
c            1.  they may be simply passed thru.
c            2.  they may be used as a flag(e.g. if dat(2) = -3.0, use
c                                    a certain set of logic).
c            3.  they may be completely ignored.
c                                     and instead redefined with new logic
c
```

2.6.6. Subroutine userfd (Calculating the Complex Load Vector for Frequency Domain Logic)

```
*deck,userfd                      USERDISTRIB
      subroutine userfd (nr,kcbrm,kpfor,ktrsur,isur,
     x cb,do,doext,aread,alenv,denswat,faclen,conac,fluidt,visc,
     x watbas,watcur,watwav,xyzup,tr,accel,puvel,u,zass,
     x forl,zsc,zsc2,pdyn,holdwv)
c *** primary function:  compute complex load vector for frequency domain logic
c                        for pipe59
c *** secondary functions: none
c     -- accessed with keyopt(12) = 2
c
c         *** Copyright ANSYS.  All Rights Reserved.
c         *** ansys, inc.
c *** Notice - This file contains ANSYS Confidential information ***
c
c  input arguments:
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c     nr       (int,sc,in)       - matrix size
c     kcbrm    (int,sc,in)       - key for reduced matrices/cable option
c     kpfor    (int,sc,in)       - keyopt for hydrodynamic printout
c     ktrsur   (int,sc,in)       - keyopt for surface treatment(unfinished)
c     isur     (int,sc,in)       - surface flag
c     cb       (dp,sc,in)        - buoyancy coefficient (real constant)
c     do       (dp,sc,in)        - outside diameter of pipe
c     doext    (dp,sc,in)        - outside diameter of insulation
c     aread    (dp,sc,in)        - area of displaced water
c     alenv    (dp,sc,in)        - length of element
c     denswat  (dp,sc,in)        - water density
c     faclen   (dp,sc,in)        - wetted fraction of pipe
c     conac    (dp,sc,in)        - added mass per unit length
c     fluidt   (dp,sc,in)        - fluid temperature
c     visc     (dp,sc,in)        - viscosity
c     watbas   (dp,ar(*),in      - water basic table
c     watcur   (dp,ar(*),in      - water current table
c     watwav   (dp,ar(*),in      - water wave table
c     xyzup    (dp,ar(3,2),in)   - updated coordinates
c     tr       (dp,ar(3,3),in)   - local to global transformation matrix
c     accel    (dp,ar(3),in)     - acceleration vector
c     puvel    (int,sc,in)       - index for velocities in u matrix
c     u        (dp,ar(nr,5),in   - displacements and velocities
c     zass     (dp,ar(nr,nr),in) - mass matrix
c     forl     (dp,ar(12),inout) - force vector in element coordinates
c     zsc      (dp,ar(nr),inout) - real load vector for frequency domain
c     zsc2     (dp,ar(nr),inout) - complex load vector for frequency domain
c
c  output arguments:
c     forl     (dp,ar(12),inout) - force vector in element coordinates
c     zsc      (dp,ar(nr),inout) - real load vector for frequency domain
c     zsc2     (dp,ar(nr),inout) - complex load vector for frequency domain
c     pdyn     (dp,ar(2),out)    - dynamic pressure
c     holdwv   (dp,ar(60),out)   - wave information held for printout
c
```

2.6.7. Function userpe (Calculating Rotation Caused by Internal Pressure)

```
*deck,userpe                       USERDISTRIB
      function userpe (prs,rvrp,angle,ex,nuxy)

c primary function:    calculate the rotation caused by internal pressure
c                      on an elbow element
c                      This function is only called by el18(pipe18)
c                                        if keyopt(5) = 1

c *** Notice - This file contains ANSYS Confidential information ***

c         *** Copyright ANSYS.  All Rights Reserved.
c         *** ansys, inc.

c     typ=int,dp,log,chr,dcp   siz=sc,ar(n)   intent=in,out,inout

c  input arguments:
c     variable (typ,siz,intent)    description
c     prs      (dp,ar(5),in)     - pressure vector
c     rvrp     (dp,ar(11),in)    - real constants(see elements manual)
c     angle    (dp,sc,in)        - subtended angle
c     ex       (dp,sc,in)        - Young's modulus
c     nuxy     (dp,sc,in)        - Poisson's ratio

c  output arguments:
c     variable (typ,siz,intent)    description
c     userpe   (dp,sc,out)       -  rotation caused by internal pressure on the
c                                     elbow element
```

2.6.8. Subroutine usrsurf116 (Modifying SURF151 and SURF152 Film Coefficients and Bulk Temperatures)

```
*deck,usrsurf116                      USERDISTRIB
      subroutine usrsurf116 (elem,ielc,center,jdim,kaxis,time,nr,u,
     x               omeg,ndat,temvel,hc,tb,temfluid,mdot,key)
c *** primary function: change element convection surface info
c     for surf151 and/or surf152 based on information from fluid116.
c     It is called by el151 and el152.
c
c         in order to activate this user programmable feature,
c         the user must have used fluid116 with keyopt(2) = 1.
c         Further, surf151 and/or surf152 must have keyopt(5) = 1 or 2
c         (include extra node).  Finally, for this routine to do anything,
c         key(1) and/or key(2) must be reset in this routine to a
c         nonzero number.  There is no usrcal control over this routine.
c
c         *** Copyright ANSYS.  All Rights Reserved.
c         *** ansys, inc.
c *** Notice - This file contains ANSYS Confidential information ***
c
c     input arguments:
c        variable (typ,siz,intent)    description
c        elem   (int,sc,in)         - element number for operation.
c        ielc   (int,ar(IELCSZ),in) - array of element type characteristics
c        center (dp,ar(3),in)       - coordinates of center of surface element
c        jdim   (int,sc,in)         - dimensionality key
c                                        1 = 2d
c                                        2 = axisymmetric
c                                        3 = 3d
c        kaxis  (int,sc,in)         - axis of rotation (keyopt(3) for el152)
c                                       (see getv116 for definition)
c        time   (dp,sc,in)          - time of current substep
c        nr     (int,sc,in)         - number of nodal temperatures
c                                         of the element
c        u      (dp,ar(nr),in)      - vector of most recent values of the
c                                      temperatures
c        omeg   (dp,sc,in)          - spin real constant (may be from table)
c        ndat   (int,sc,in)         - number of data points per element
c        hc     (dp,ar(ndat),inout) - film coefficients
c                                      (has input values for each corner
c                                      of element)
c        tb     (dp,ar(ndat),in) - bulk temperature
c                                     (has input values for each corner
c                                     of element)
c        temfluid (dp,sc,in)        - temp of fluid at surf151/152 centroid
c                                   - when using kyop5 = 1 or 2
c        mdot (dp,sc,in)            - mass flow rate of fluid when using
c                                   - kyop5 = 2 ( 0 otherwise )
c
c     output arguments:
c        variable (typ,siz,intent)    description
c        temvel (dp,sc,out)         - user defined bulk temperature in excess of
c                                      fluid node temperature
c        hc     (dp,ar(ndat),inout) - film coefficients
c                                      (defines input values for each corner
c                                      of element)
c        key    (int,ar(2),out)     - key if to use this logic
c                                      key(1) = 0 = no new film coefficient
c                                      key(1) = 1 = define new film coefficient
c                                      key(2) = 0 do not use any temvel
c                                             = 1 use constant temvel
c                                             = 2 use bilinear variation
c                                                 of temvel by
c                                                 redefining tb array
c
c         this routine is called during each substep of each load step.
c         it is called for each equilibrium iteration.
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c         it is called once per element.  it is called only during the heat
c         flow load vector formulation stage, and not during the heat flow
c         evaluation stage.
c         the call to get the standard ansys input convection surfaces
c         is made just before entering this routine, so this information is
c         available to be modified, if desired.
c
c         This routine may be thought of as a specialized version of usercv.
c         Indeed, el151 and el152 also call usercv.  Either (or both, rarely)
c         could be used.
c
c         velocity-dependent film coefficients and bulk temperatures can
c         be computed by using the velocities and other information from
c         fluid116.
c         Details of this procedure are:
c            -- SURF151 or SURF152 are 'pasted' onto the actual solid model.
c            -- flow rate is input to or is computed by FLUID116,
c                 with KEYOPT(2) = 1
c            -- flow rate may be a function of time
c            -- the user defines nodes on the FLUID116 network to be the same
c                 nodes as the 'extra' nodes of SURF151 or SURF152.  If more
c                 than one FLUID116 element is attached to one of these nodes,
c                 the velocities are averaged.
c            -- SURF151 or SURF152 calls this routine, indirectly, to compute
c                 the film coefficient and bulk temperature.  This routine,
c                 in turn, gets the average velocity at the 'extra' node
c                 using 'getv116', as shown below.  Other quantities brought
c                 in by getv116 are also averaged.
```

2.6.9. Subroutine User116Cond (Calculating the Conductance Coefficient for FLUID116)

```
*deck,User116Cond                       USERDISTRIB
      subroutine User116Cond(elem,prop,rvr,aleng,re,fric,uptot,uttot,
     x    bco)
c primary function:  compute bc for conductance coefficient for fluid116

c *** Notice - This file contains ANSYS Confidential information ***
c
c         *** Copyright ANSYS.  All Rights Reserved.
c         *** ansys, inc.

c  input arguments:
c     elem     (int,sc,in)       - element number
c     prop     (dp,ar(4),in)     - material property vector
c                                    order is:  dens,visc,kxx,c
c     rvr      (dp,ar(24),in)    - real constant vector
c     aleng    (dp,sc,in)        - element length
c     re       (dp,sc,in)        - reynold's number
c     fric     (dp,sc,in)        - friction factor
c     uptot    (dp,ar(2),in      - nodal pressure values from previous iteration
c     uttot    (dp,ar(4),in      - nodal temperature values from prev iteration
c     bco      (dp,sc,inout)     - the conductance coefficient from TB,fcon

c  output arguments:
c     bco      (dp,sc,inout)     - the desired conductance coefficient
```

2.6.10. Subroutine User116Hf (Calculating the Film Coefficient for FLUID116)

```
*deck,User116Hf                       USERDISTRIB
```

```
      subroutine User116Hf (elem,prop,rvr,aleng,re,uptot,uttot,hf)
c primary function:  compute hf for film coefficient for fluid116

c *** Notice - This file contains ANSYS Confidential information ***
c
c         *** Copyright ANSYS.  All Rights Reserved.
c         *** ansys, inc.

c  input arguments:
c     elem     (int,sc,in)       - element number
c     prop     (dp,ar(4),inout)  - material property vector
c                                    order is:  dens,visc,kxx,c
c     rvr      (dp,ar(18),in)    - real constant vector
c     aleng    (dp,sc,in)        - element length
c     re       (dp,sc,in)        - reynold's number
c     uptot    (dp,ar(2),in      - nodal pressure values from previous iteration
c     uttot    (dp,ar(4),in      - nodal temperature values from prevs iteration
c     hf       (dp,sc,inout)     - the film coefficient from TB,hflm
c                                - as a function of temp and velocity

c  output arguments:
c     hf       (dp,sc,inout)     - the desired film coefficient
```

2.6.11. Subroutine userPartVelAcc (Calculating Particle Velocities and Accelerations of Ocean Waves)

The `userPartVelAcc` subroutine is the primary component of the API for inputting your own wave and current information. The API supports the hydrodynamic capability available with line elements (such as LINK180, BEAM188, BEAM189, PIPE288, and PIPE289). The `userPartVelAcc` subroutine works with the following subroutines:

- `userPartVelAccSetup` (p. 261), which initializes the data for use by `userPartVelAcc`, and
- `userWavHt` (p. 262), which calculates the wave height for a user-defined wave.

For your convenience, two I/O service subroutines are called by the `userPartVelAcc` subroutine: `wvhybl` (p. 263) and `wvargu` (p. 263).

```
*deck,userPartVelAcc                      USERDISTRIB
      subroutine userPartVelAcc (elemId,domInt,xyzg,doIns,depth,denswat,
     x                           ncm, pCur,watcur,
     x                           nw,  pWav,watwav, timval,
     x                           argu,eta,vxyz,axyz,ar,pdynam)
c        ---- accessed only if kwav .ge. 101 -----
c   ***** primary function:  compute particle velocities and accelerations
c                           due to waves and current
c   ***** secondary function:  compute dynamic pressures
c
c         *** Copyright ANSYS.  All Rights Reserved.
c         *** ansys, inc.
c *** Notice - This file contains ANSYS Confidential information ***
c
c  input arguments:
c     elemId   (int,sc,in)       - element id
c     domInt   (int,sc,in)       - integration point number
c     xyzg     (dp,ar(3),in)     - coordinates of point of interest
c     doIns    (dp,sc,in)        - outside diameter with insulation
c     depth    (dp,sc,in)        - water depth
c     denswat  (dp,sc,in)        - water density
c     ncm      (int,sc,in)       - number of current measurements
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c     pCur     (int,sc,in)       - pointer in current table (= 30 at 12.0)
c                                    i.e. first item is at watcur(pCur+1)
c     watcur   (dp,ar(*),in)     - water current table
c                                          ic = current reading number
c                                   watcur( 6) = ncm = number of current measurements
c                                   watcur(pCur + (ic-1)*6 + 1) = Z Coor
c                                   watcur(pCur + (ic-1)*6 + 2) = Velocity
c                                   watcur(pCur + (ic-1)*6 + 3) = Angle
c                                   watcur(pCur + (ic-1)*6 + 4) = Temperature
c                                   watcur(pCur + (ic-1)*6 + 5) = Spare
c                                   watcur(pCur + (ic-1)*6 + 6) = Spare
c     nw       (dp,sc,in)        - number of wave components
c     pWav     (int,sc,in)       - pointer to wave table (= 30 at 12.0)
c     watwav   (dp,ar(*),in)     - water wave table
c                                   watwav( 6) = nw = number of wave components
c                                   watwav(11) = KWAVE  (kwav)
c                                   watwav(12) = THETA
c                                   watwav(13) = WAVLOC (kpeak)
c                                   watwav(14) = KCRC
c                                   watwav(15) = KMF
c                                   watwav(16) = PRKEY
c                                          iw = wave number
c                                   watwav(pWav + (iw-1)*6 + 1) = Wave Height
c                                   watwav(pWav + (iw-1)*6 + 2) = Period
c                                   watwav(pWav + (iw-1)*6 + 3) = Phase Shift
c                                   watwav(pWav + (iw-1)*6 + 4) = Wave Length
c                                   watwav(pWav + (iw-1)*6 + 5) = Spare
c                                   watwav(pWav + (iw-1)*6 + 6) = Spare
c     timval   (dp,sc,in)        - current time value
c
c  output arguments:
c       While the below 7 arguments are output, they can also
c       be used as input, based on other ANSYS input.
c     argu     (dp,sc,out)       - position in wave (radians) (passed out only for output)
c     eta      (dp,sc,out)       - total wave height
c     vxyz     (dp,ar(3),out)    - particle velocities
c     axyz     (dp,ar(3),out)    - particle accelerations
c     ar       (dp,sc,out)       - radial particle acceleration
c     pdynam   (dp,sc,out)       - dynamic pressure head
c
c  local variable
c     phead    (dp,sc,out)       - pressure head
c
```

2.6.11.1. Subroutine userPartVelAccSetup (Initializing Data for Use by the userPartVelAcc Subroutine)

This subroutine initializes the data for the `userPartVelAcc` (p. 260) subroutine.

```
*deck,userPartVelAccSetup                      USERDISTRIB
      subroutine userPartVelAccSetup ( kch,ptr_Ocean,
     x                                 nsize,nsizec,nsizew,
     x                                 dWork,dWorkC,dWorkW,
     x                                 rkd,wvmax )
c        ---- accessed only if kwav .ge. 101 -----
c *** primary function: set up and checking of user wave (and current) theory
c *** secondary functions: none
c *** Notice - This file contains ANSYS Confidential information ***
c Copyright ANSYS.  All Rights Reserved.
c
c
c  input arguments:
c     kch      (int,sc,in)       - key for checking or defaulting (not used by PIPE288)
c     ptr_Ocean (int,sc,in)      - storage offset
c     nsize    (int,sc,in)       - size of ocean basic data
```

```
c     nsizec   (int,sc,in)       - size of ocean current data
c     nsizew   (int,sc,in)       - size of ocean wave data
c     dWork    (dp,ar(*),inout)  - raw ocean basic data (dWork = watbas)
c                                  watbas( 6) = nReN = number of Reynold's numbers
c                                  watbas(11) = DEPTH
c                                  watbas(12) = MATOC
c                                  watbas(13) = KFLOOD
c                                  watbas(14) = Ci
c                                  watbas(15) = Cb
c                                        pBas = 30 (at Rev 12.0) (to be added to argument list)
c                                          ir = Reynold's number number
c                                  watbas(pBas + (ir-1)*9 + 1) = RE
c                                  watbas(pBas + (ir-1)*9 + 2) = CDy
c                                  watbas(pBas + (ir-1)*9 + 3) = CDz
c                                  watbas(pBas + (ir-1)*9 + 4) = CT
c                                  watbas(pBas + (ir-1)*9 + 5) = CMy
c                                  watbas(pBas + (ir-1)*9 + 6) = CMz
c     dWorkC   (dp,ar(*),inout)  - raw ocean current data (dWorkC = watcur)
c     dWorkW   (dp,ar(*),inout)  - raw ocean wave    data (dworkW = watwav)
c                                     - see userPartVelAcc.F for details for watcur and watwav
c
c  output arguments:
c     dWork    (dp,ar(*),inout)  - adjusted ocean basic data
c     dWorkC   (dp,ar(*),inout)  - adjusted ocean current data
c     dWorkW   (dp,ar(*),inout)  - adjusted ocean wave data
c     rkd      (dp,sc,out)       - value of k*d
c     wvmax    (dp,sc,out)       - total wave height
```

2.6.11.2. Subroutine userWavHt

The `userWavHt` subroutine calculates the wave height of a user-defined wave for the `userPartVelAcc` (p. 260) subroutine.

```
*deck,userWavHt                       USERDISTRIB
      subroutine userWavHt (xyzg,doext,depth,nw,pWav,watwav,timval,
     &                      eta,etadot)
c        ---- accessed only if kwave .ge. 101 -----
c     *** primary function: calculate wave height for user wave
c     ***                        over point at xyzg of the element
c     *** secondary functions:  none
c
c *** Notice - This file contains ANSYS Confidential information ***
c Copyright ANSYS.  All Rights Reserved.
c
c  input arguments:
c     xyzg     (dp,ar(3),in)       - updated coordinates of point of interest in
c     doext    (dp,sc,in)          - outside diameter with insulation
c                                    if timval<0.0, argu = doext
c     depth    (dp,sc,in)          - water depth
c     nw       (int,sc,in)         - number of waves
c     pWav     (int,sc,in)         - pointer to wave table
c     watwav   (dp,ar(*),in)       - water wave table
c     timval   (dp,sc,in)          - current time value
c                                    if timval < 0.0
c                                       pass directly in doext position
c                                       (used for stream function only)
c                                    else compute value in wvargu
c
c  output arguments:
c     eta      (dp,sc,out)         - wave height
c     etadot   (dp,sc,out)         - time derivative of wave height
c
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.6.11.3. Subroutine wvhybl

The wvhybl subroutine computes the ratio of two hyperbolic functions and is intended for use with wave loading. It is a utility subroutine called by the userPartVelAcc (p. 260) subroutine.

```
*deck,wvhybl
      function wvhybl (kclass,x,y)
c *** primary function: to compute the ratio of two hyperbolic functions,
c                       specialized to the needs of wave loading.
c                       The options are as given with kclass below.
c                       Further, only positive values of x and y are used
c
c *** secondary functions: none
c
c *** Notice - This file contains ANSYS Confidential information ***
c
c  input arguments:
c     variable (typ,siz,intent)    description
c     kclass   (int,sc,in)       - 0 - cosh(x)/cosh(y)
c                                - 1 - sinh(x)/cosh(y)
c                                - 2 - cosh(x)/sinh(y)
c                                - 3 - sinh(x)/sinh(y)
c     x        (dp,sc,in)        - argument of numerator
c     y        (dp,sc,in)        - argument of denominator
c
c  output arguments:
c     variable (typ,siz,intent)    description
c     wvhybl   (dp,sc,out)       - resulting fraction
c
```

2.6.11.4. Subroutine wvargu

The wvargu subroutine computes the appropriate position with regard to the wave. It is a utility subroutine called by the userPartVelAcc (p. 260) subroutine.

```
*deck,wvargu
      function wvargu (kpeak,kmf,wavdat,timval,r,doext)
c     *** primary function: to find appropriate position wrt wave
c     *** secondary functions:  none
c
c *** Notice - This file contains ANSYS Confidential information ***
c
c  input arguments:
c     kpeak    (int,sc,in)       - keyopt for when peak effect occurs
c     kmf      (int,sc,in)       - key for maccamy-fuchs adjustment
c     wavdat   (dp,ar(6),in)     - wave data (from water wave table)
c                                        wavdat(1) = wave height(not used)
c                                        wavdat(2) = period
c                                        wavdat(3) = phase shift
c                                        wavdat(4) = wave length
c     timval   (dp,sc,in)        - current time value
c     r        (dp,sc,in)        - radial location of point of interest
c     doext    (dp,sc,in)        - effective outside diameter of pipe
c
c  output arguments:
c     wvargu   (dp,sc,out)       - wave position(as determined by the argument)
c                                  output in radians
c
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.6.12. Subroutine userPanelHydFor (Calculating Panel Loads Caused by Ocean Loading)

The `userPanelHydFor` subroutine applies loads and other effects onto SURF154 surface elements. This capability is accessed via KEYOPT(8) of SURF154, together with data read in via the `userOceanRead (p. 264)` subroutine.

```
*deck,userPanelHydFor                      USERDISTRIB
      subroutine userPanelHydFor (kPOcean, elemId, intPnt,
     x depth, denswat,
     x ncm, pCur, watcur,
     x nw , pWav, watwav,
     x xyzupp, vn,
     x presoc,admsoc)

c        ---- accessed only if kwave on the OCDATA command .ge. 101 -----
c primary function:    Get pressure loading on panel
c secondary functions: Get hydrodynamic mass on panel
c       load is applied on SURF154 with keyopt(8)

c *** Notice - This file contains ANSYS Confidential information ***
c Copyright ANSYS.  All Rights Reserved.

c  parameter definition include files:
```

2.6.12.1. Subroutine userOceanRead

The `userOceanRead` subroutine reads in ocean data to be used by the `userPanelHydFor (p. 264)` subroutine.

```
*deck,userOceanRead                        USERDISTRIB

      subroutine userOceanRead (iott,kpr,fUnitNo,iOption,
     x                          pdWaveData,lenWavDat)

c        ---- accessed only if kwave on the OCDATA command .ge. 101 -----
c Primary Function: read in ocean file for later use
c Secondary Functions:
c
c -----------------
c Notice:
c ------
C    This routine contains ANSYS, Inc. confidential information
c Copyright ANSYS.  All Rights Reserved.
c -----------------
c  input arguments:
c     iott         (int,sc,in)     output unit number, based on then /OUT command
c     kpr          (log,sc,in)     print flag, based on the /NOPR command
c     fUnitNo      (int,sc,in)     file unit number, based on the command
c                                  OCREAD,file,ext,dir
c     iOption      (int,sc,in)     integer from the command line, based on
c                                  OCREAD,file,ext,dir,iOption
c     pdWaveData   (ptr,sc,out)    pointer to wave data array
c     lenWavDat    (int,sc,out)    length of wave data
c                                  0 = an error, no wave data is stored
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.7. Subroutines for Sharing Data Between User Routines

In Mechanical APDL running under Windows, each user routine is built into a separate dynamic link library (DLL). To share data, functions and data must be explicitly exported and imported. The following subroutines enable you to share data between user routines via common-block variables:

For more information, see Sharing Data Between User Routines (p. 137).

2.7.1. Subroutine userdata (Store Common Block Functionality and Data)

```
*deck,userdata                       USERDISTRIB
      function getusercmvals(iloc,sz,outdata)
c!DEC$ ATTRIBUTES DLLEXPORT :: getusercmvals
#include "usercm.inc"
      integer         iloc,sz,getusercmvals
      double precision outdata(*)
      if ( iloc.lt.1.or.iloc+sz.gt.userdatsz) then
          getusercmvals = 0
      else
          outdata(1:sz)= userdata(iloc:iloc+sz)
          getusercmvals = 1
      endif
      return
      end

      function setusercmvals(iloc,sz,indata)
c!DEC$ ATTRIBUTES DLLEXPORT :: setusercmvals
#include "usercm.inc"
      integer         iloc,sz,setusercmvals
      double precision indata(*)
      if ( iloc.lt.1.or.iloc+sz.gt.userdatsz) then
          setusercmvals = 0
      else
          userdata(iloc:iloc+sz) = indata(1:sz)
          setusercmvals = 1
      endif

      return
      end

      subroutine initusercmvals(arraysz)
c!DEC$ ATTRIBUTES DLLEXPORT :: initusercmvals
#include "impcom.inc"
#include "ansysdef.inc"
#include "usercm.inc"
      integer arraysz
      external  fAnsMemAlloc
      PTRFTN   fAnsMemAlloc
      character*16   memlabel
      memlabel = 'userdat'
      userdatptr = fAnsMemAlloc(arraysz,MEM_DOUBLE,memlabel)
      userdatsz = arraysz
      return
      end

      subroutine freeusercmvals()
c!DEC$ ATTRIBUTES DLLEXPORT :: freeusercmvals
#include "usercm.inc"
      external fAnsMemFree
      call fAnsMemFree(userdatptr)
      userdatptr = 0
      return
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
      end

      function getusercmvalsz()
c!DEC$ ATTRIBUTES DLLEXPORT :: getusercmvalsz
#include "usercm.inc"
      integer getusercmvalsz
      getusercmvalsz = userdatsz
      return
      end
```

2.7.2. Subroutine usercm.inc (Add Common Block Variables)

Use this subroutine with `userdata` to add more common blocks.

```
*comdeck,usercm USERDISTRIB
c!DEC$ ATTRIBUTES DLLEXPORT :: /usercm/
        common /usercm/  userdatsz,userdatptr
        double precision userdata(*)
        pointer (userdatptr,userdata)
        integer userdatsz
```

2.8. Running Mechanical APDL as a Subroutine

To call the Mechanical APDL program, use the following:

```
program ansys
```

For multiple calls to subroutine `ansys`, you must open and close standard input in the calling subroutine. (Usually, input and output are FORTRAN units 5 and 6, respectively.) The calling subroutine cannot use the database access subroutines; however, other user-programmable features can use the database access subroutines freely.

There may be times when Mechanical APDL exits abnormally. Check the `file.err` file to see if Mechanical APDL wrote an exit code to the file before ending. These error codes may help you to understand what caused the abnormal program exit:

Table 2.1: ANSYS Exit Codes

Code	Explanation	Code	Explanation
0	Normal Exit	14	XOX Error
1	Stack Error	15	Fatal Error
2	Stack Error	16	Possible Full Disk
3	Stack Error	17	Possible Corrupted or Missing File
4	Stack Error	18	Possible Corrupted DB File
5	Command Line Argument Error	21	Authorized Code Section Entered
6	Accounting File Error	25	Unable to Open X11 Server
7	Auth File Verification Error	30	Quit Signal

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Code	Explanation	Code	Explanation
8	Error in Mechanical APDL or End-of-run	31	Failure to Get Signal
11	User Routine Error	>32	System-dependent Error
12	Macro STOP Command		

2.9. Defining Your Own Commands

Ansys, Inc. provides a set of user subroutines named `user01` through `user10` for defining custom commands:

To define a custom command:

1. Insert the code for the functions you want to perform into subroutine `user01` (or `user02`, etc.).
2. Link the subroutine into the program.
3. Issue the command **/UCMD** to define a name for a custom command that calls and executes your subroutine. Use the command format shown below:

```
/UCMD,Cmd,SRNUM
```

Cmd

The name for your new command. It can contain any number of characters, but only the first four are significant. The name you specify can not conflict with the name of any command or the names of any other commands or macros.

SRNUM

The number of the subroutine your command should call; that is, a value between 01 and 10. For example, suppose that you create and link in a user subroutine for a parabolic distribution of pressure, and you name that subroutine `user01`. Issuing the command shown below creates a new command, PARB, that when issued calls your parabolic pressure distribution subroutine:

```
/UCMD,PARB,1
```

To make these "custom command" subroutines available in all your sessions, include the **/UCMD** commands in your start-up file (START**.**ANS).

You also can use **/UCMD** to remove a custom command. To do so, simply use a blank value for *Cmd*, as shown below:

```
/UCMD,,1
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This command removes the PARB command. To list all user-defined command names, issue the command **/UCMD**,STAT.

2.9.1. Function user01

```
*deck,user01                     USERDISTRIB
      function  user01()
c *** primary function:    user routine number  01

c         *** Copyright ANSYS.  All Rights Reserved.
c         *** ansys, inc.
c *** Notice - This file contains ANSYS Confidential information ***

c  /*****************************************************************\
c  | this is a user routine that may be used by users to include their |
c  | special coding.  accesss to this routine is by the command usr1.  |
c  | usr1 may be changed by the user using the command /ucmd.  the     |
c  | user may then use this routine to call his/her special routines.  |
c  | ansys routines to access information in the ansys database may be |
c  | found in the "ansys programmer's manual", available from ansys,inc|
c  | see user02 for a simple example usage.                            |
C  | routines user03 to user10 are also available.                     |
C  \*****************************************************************/

c  input arguments:  none

c  output arguments:
c     user01   (int,sc,out)      - result code (should be zero)
c                                   (which is ignored for now)

c    ************************************************************
c    Functions for accessing data on the command line
c    integer function  intinfun(iField) - gets an integer from field iField
c    double precision function dpinfun(iField) - gets double precision
c    character*4 ch4infun(iField) - gets (upper case) 4 characters
c    character*8 ch8infun(iField) - gets (mixed case) 8 characters
c    character*32  ch32infun(iField) - gets (mixed case) 32 characters
c    ************************************************************
c
#include "impcom.inc"
#include "ansysdef.inc"

      external  wrinqr
      integer   wrinqr

      integer  user01, iott

      iott = wrinqr(2)

c          *****  USER'S CODE IS INSERTED HERE *****
      write (iott,2000)
 2000 format (//' *****  CALL TO ANSYS,INC DUMMY USER01  *****'//)

c          *****  do not return this result code in a real user routine
      user01 = -654321
c         *****  instead return a zero   *****
c     user01 = 0

      return
      end
```

2.9.2. Function user02 (Demonstrates Offsetting Selected Nodes)

```
*deck,user02                     USERDISTRIB
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
      function  user02()
c *** primary function:    user routine number  02
c      --- This demonstration offsets selected nodes with the command:
c              usr2,dx,dy,dz

c         *** Copyright ANSYS.  All Rights Reserved.
c         *** ansys, inc.
c *** Notice - This file contains ANSYS Confidential information ***

c  /*****************************************************************\
c  | see user01 for additional information on user routines          |
c  \*****************************************************************/

c  input arguments:  none

c  output arguments:
c     user02   (int,sc,out)      - result code (should be zero)
c                                  (which is ignored for now)

c    ************************************************************
c    Functions for accessing data on the command line
c    integer function  intinfun(iField) - gets an integer from field iField
c    double precision function dpinfun(iField) - gets double precision
c    character*4 ch4infun(iField) - gets (upper case) 4 characters
c    character*8 ch8infun(iField) - gets (mixed case) 8 characters
c    character*32  ch32infun(iField) - gets (mixed case) 32 characters
c    ************************************************************
c
#include "impcom.inc"
#include "ansysdef.inc"

      external  wrinqr,ndinqr,ndgxyz,ndpxyz,erhandler, dpinfun
      integer   wrinqr,ndinqr,ndgxyz
      double precision  dpinfun

      integer  user02, iott, maxnp, i ,ksel
      double precision  xyz(3), offset(3)


      maxnp = ndinqr(0,DB_MAXDEFINED)

c          *****  get the desired offsets from the command line  *****
      offset(1) = dpinfun(2)
      offset(2) = dpinfun(3)
      offset(3) = dpinfun(4)

      do i = 1,maxnp
          ksel = ndgxyz (i,xyz(1))
          if (ksel .eq. 1) then
             xyz(1) = xyz(1) + offset(1)
             xyz(2) = xyz(2) + offset(2)
             xyz(3) = xyz(3) + offset(3)
             call ndpxyz (i,xyz(1))
          endif
      enddo

c          *****  write to output file  *****
      iott = wrinqr(WR_OUTPUT)
      write (iott,2000)
 2000 format (/'  NODE OFFSET COMPLETE  '/)

c         *****  write to GUI window  *****
      call erhandler ('user02',3000,
     x             2,'NODE OFFSET COMPLETE',0.0d0,' ')

c          *****  required return value  *****
      user02 = 0

      return
      end
```

2.9.3. Function user03 (Demonstrates Using Memory)

```
*deck,user03                      USERDISTRIB
      function  user03()
c *** primary function:    user routine number 03. Gives example of
c                          ANSYS Memory usage

c         *** Copyright ANSYS.  All Rights Reserved.
c         *** ansys, inc.
c *** Notice - This file contains ANSYS Confidential information ***

c  /*****************************************************************\
c  | see user01 for additional information on user routines          |
c  \*****************************************************************/

c  input arguments:  none

c  output arguments:
c     user03   (int,sc,out)      - result code (should be zero)
c                                  (which is ignored for now)

c    ************************************************************
c    Functions for accessing data on the command line
c    integer function  intinfun(iField) - gets an integer from field iField
c    double precision function dpinfun(iField) - gets double precision
c    character*4 ch4infun(iField) - gets (upper case) 4 characters
c    character*8 ch8infun(iField) - gets (mixed case) 8 characters
c    character*32  ch32infun(iField) - gets (mixed case) 32 characters
c    ************************************************************

#include "impcom.inc"
#include "ansysdef.inc"

      external  wrinqr, ndinqr, ndgxyz, ndnext, fAnsMemAlloc,
     x          fAnsMemFree,erhandler, parreturn, parstatus
      integer   wrinqr, ndinqr, ndgxyz, ndnext
      PTRFTN    fAnsMemAlloc

      integer   user03, iott, i, ksel, numnp, node, istat
      double precision  xyz(3), xmean, ymean, zmean, stdxyz(3),
     x          sodx, sody, sodz

c  pointers:
      pointer (pdXnodeL,Xnode)
      pointer (pdYnodeL,Ynode)
      pointer (pdZnodeL,Znode)
      double precision  Xnode(*), Ynode(*), Znode(*)


c     *** Get nodal xyz locations and calculate standard deviation of
c     *** x coordinates, y coordinates, & z coordinates


c     *** get number of currently selected nodes
      numnp = ndinqr(0,DB_NUMSELECTED)

      istat = 1
      if (numnp .le. 0) go to 999

c     *** allocate memory for x, y, & z coordinates of nodes
      pdXnodeL = fAnsMemAlloc(numnp,MEM_DOUBLE,'XCoords ')
      pdYnodeL = fAnsMemAlloc(numnp,MEM_DOUBLE,'YCoords ')
      pdZnodeL = fAnsMemAlloc(numnp,MEM_DOUBLE,'ZCoords ')

c     *** loop through all selected nodes
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
      i = 1
      node = 0
      xmean = 0.0d0
      ymean = 0.0d0
      zmean = 0.0d0

 10   node = ndnext(node)

      if (node .gt. 0) then

c         *** get xyz coordinates
          ksel = ndgxyz(node,xyz(1))

c         *** store this node's xyz coordinates
          Xnode(i) = xyz(1)
          Ynode(i) = xyz(2)
          Znode(i) = xyz(3)

c         *** while we're looping, accumulate sums to calculate means
          xmean = xmean + xyz(1)
          ymean = ymean + xyz(2)
          zmean = zmean + xyz(3)

c         *** increment index
          i = i + 1

c         *** loop back up for next selected node
          goto 10

      endif

c     *** node = 0, at the end of node list

c     *** calculate mean of xyz coordinates
      xmean = xmean / numnp
      ymean = ymean / numnp
      zmean = zmean / numnp

c     *** calculate standard deviation for xyz coordinates
      sodx = 0
      sody = 0
      sodz = 0
      do  i = 1, numnp
         sodx = sodx + (Xnode(i) - xmean)**2
         sody = sody + (Ynode(i) - ymean)**2
         sodz = sodz + (Znode(i) - zmean)**2
      enddo

      stdxyz(1) = sqrt(sodx / (numnp-1))
      stdxyz(2) = sqrt(sody / (numnp-1))
      stdxyz(3) = sqrt(sodz / (numnp-1))

c     *****  write to output file  *****
      iott = wrinqr(WR_OUTPUT)
      write (iott,2000) xmean,ymean,zmean,
     x                  stdxyz(1),stdxyz(2),stdxyz(3)
 2000 format (/' MEAN FOR X COORDINATES:',G12.5/
     x         ' MEAN FOR Y COORDINATES:',G12.5/
     x         ' MEAN FOR Z COORDINATES:',G12.5/
     x         ' STD  FOR X COORDINATES:',G12.5/
     x         ' STD  FOR Y COORDINATES:',G12.5/
     x         ' STD  FOR Z COORDINATES:',G12.5)

c     *****  write to GUI window  *****
      call erhandler ('user03',5000,2,
     x 'STD  FOR X COORDINATES: %G %/
     x  STD  FOR Y COORDINATES: %G %/
     x  STD  FOR Z COORDINATES: %G',stdxyz(1),' ')

c      *****  set _STATUS to 0 for success  *****
      istat = 0
```

```
c     *** release dynamically allocated memory
      call fAnsMemFree (pdZnodeL)
      call fAnsMemFree (pdYnodeL)
      call fAnsMemFree (pdXnodeL)

c     *****  required return value  *****
 999  user03 = 0

c     *****  set _RETURN  to number of nodes processed  *****
      call parreturn (dble(numnp))

c     *****  set _STATUS for success (0) or no nodes (1)  *****
      call parstatus (istat)

      return
      end
```

2.9.4. Function user04

```
*deck,user04                      USERDISTRIB
      function  user04()
c *** primary function:   user routine number  04; demonstrates gettting a
c                         list of nodes attached to a keypoint, line, or area

c         *** Copyright ANSYS.  All Rights Reserved.
c         *** ansys, inc.
c *** Notice - This file contains ANSYS Confidential information ***

c  /*****************************************************************\
c  | see user01 for additional information on user routines          |
c  \*****************************************************************/

c  input arguments:  none

c  output arguments:
c     user04   (int,sc,out)      - result code (should be zero)
c                                  (which is ignored for now)

c    ************************************************************
c    Functions for accessing data on the command line
c    integer function  intinfun(iField) - gets an integer from field iField
c    double precision function dpinfun(iField) - gets double precision
c    character*4 ch4infun(iField) - gets (upper case) 4 characters
c    character*8 ch8infun(iField) - gets (mixed case) 8 characters
c    character*32  ch32infun(iField) - gets (mixed case) 32 characters
c    ************************************************************
c
#include "impcom.inc"
#include "ansysdef.inc"


      external  ndkpnt

      external  wrinqr, ndline, ndarea, intinfun
      integer   wrinqr, ndline, ndarea, intinfun
      external     ch4infun
      character*4  ch4infun

      integer  user04,   iott, listk(20),listl(20),lista(20), listin(1),
     x           i, num,ktype, nkpnts, nlines, nareas
      character*4  type, lab2


      iott = wrinqr (WR_OUTPUT)

c     --- setup with:  /UCMD,GNSME,4
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c        !gnsme,group,num,type
c        ! group = kp, ln, or ar
c        ! num   = entity number of kp, ln, or ar
c        ! type = interior, or all
c         ---- see input deck dv-5805s

      lab2 = ch4infun(2)
      write (iott,2010) lab2
 2010 format(/' group name (type of entity) = ',a4)

      num = intinfun(3)
      write (iott,2020) num
 2020 format (' entity number =',i4)
      listin(1) = num

      if (lab2 .ne. 'KP  ' ) then
         type = ch4infun(4)
         if (type .eq. 'INTE') then
            write (iott,2030)
 2030       format (' interior nodes only ')
            ktype = 0
         elseif (type .eq. 'ALL ') then
            write (iott,2040)
 2040       format (' all (interior and edge/end) nodes ')
            ktype = 1
         else
            write (iott,2050)
 2050       format ('Only INTE or ALL are acceptable in last field',
     x       ' on user-written gnsme command')
         endif
      endif

      if (lab2 .eq. 'KP  ' ) then
         nkpnts = 0
         call ndkpnt (1,listin(1),nkpnts,listk(1))
         write (iott,2110) nkpnts
 2110    format (' number of nodes on keypoint = ',i4)
         write (iott,2115) (listk(i),i=1,nkpnts)
 2115    format (' node on keypoint = ',i4)

      elseif (lab2 .eq. 'LN  ' ) then
         nlines = ndline (num,ktype,listl(1))
         write (iott,2120) nlines
 2120    format (' number of nodes on line = ',i4)
         write (iott,2125) (listl(i),i=1,nlines)
 2125    format (' list of nodes on line'/(3x,i4))

      elseif (lab2 .eq. 'AR  ' ) then
         nareas = ndarea (num,ktype,lista(1))
         write (iott,2130)  nareas
 2130    format (' number of nodes on area = ',i4)
         write (iott,2135) (lista(i),i=1,nareas)
 2135    format (' list of nodes on area'/(3x,i4))

      else
         write (iott,2150)
 2150    format (' Only KP, LN, or AR are acceptable on user-written ',
     x    'gnsme command')
      endif

      user04 = 0

      return
      end
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.9.5. Functions user05 through user10

The source code for user subroutines `user05`, `user06`, `user07`, `user08`, `user09`, and `user10` is identical to function `user01` shown above.

2.10. Support Subroutines

The following subroutines are available as a convenience for general applications:

2.10.1. Function GetRForce (Getting Nodal Reaction Force Values)

```
*deck,GetRForce
      function GetRForce (Node,Label,Value)
c primary function:    Get the K * u - F at a node from the rfsum vector.
c                       warning: This routine is called after the elements
c                       are formed, but before solution.  Therefore,
c                       F is from the current iteration, but
c                       u is from the previous iteration.  At convergence,
c                       this difference will have little effect.
c                       The computations are done immediately after the
c                       call to UElMatx.
c               Use the RFSUM command to ask for the summation.
c               Use *GET,Parm,NODE,num,RF,DOFLAB to access the reaction
c                         sum from the command line.
c secondary functions: Return pointer for fast access

c object/library:  usr

c *** Notice - This file contains ANSYS Confidential information ***
c     Prolog is not CONFIDENTIAL INFORMATION

c  input arguments:
c     variable (typ,siz,intent)    description
c     Node     (int,sc,in)       - Node Number (User)
c     Label    (ch*4,sc,in)      - DOF Label (Upper Case)
c                                  'UX  ','UY  ','TEMP','VOLT','ROTY', etc

c  output arguments:
c     GetRForce (int,func,out)   - status/pointer
c                                  = 0 - data not valid
c                                  > 0 - Rfsum pointer to data for fast access
c                                         see comments below
c     Value    (dp,sc,out)       - Solution value for Node,Label
c                                   All results are in the nodal coordinate
c                                   system

c  example usage:

c       external  GetRForce
c       integer   GetRForce, ptr, Node2
c       double precision  Value
c #include "handlecom.inc"   (if Value = Rfsum(ptr) form is to be used)

c       ptr  = GetRForce (Node2,'UY  ',Value)
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c     later...
c       Value = Rfsum(ptr)
c   directionID is used to translate label into corresponding position in dislab's position
```

2.10.2. Function GetStackDisp (Getting Current Displacement Values)

```
*deck,GetStackDisp
      function GetStackDisp (Node,Label,Value)

c primary function:    Get the displacement at a node from the disp vector
c secondary functions: Return pointer for fast access

c object/library:  usr

c *** Notice - This file contains ANSYS Confidential information ***
c     Prolog is not CONFIDENTIAL INFORMATION

c     typ=int,dp,log,chr,dcp   siz=sc,ar(n)   intent=in,out,inout

c  input arguments:
c     variable (typ,siz,intent)    description
c     Node     (int,sc,in)       - Node Number (User)
c     Label    (ch*4,sc,in)      - DOF Label (Upper Case)
c                                  'UX  ','UY  ','TEMP','VOLT','ROTY', etc

c  output arguments:
c     variable (typ,siz,intent)    description
c     GetStackDisp (int,sc,out)  - status/pointer
c                                  = 0 - data not valid
c                                  > 0 - UDisp pointer to data for fast access
c                                        see comments below
c     Value    (dp,sc,out)       - Solution value for Node,Label

c  example usage:

c       external  GetStackDisp
c#include "handlecom.inc"  (only if UDisp(ptr) form is used
c       integer   GetStackDisp, ptr, Node2
c       double precision  Value

c       ptr  = GetStackDisp (Node2,'UY  ',Value)

c     later...
c       Value = UDisp(ptr)
```

2.10.3. Subroutine ElResultStrt (Getting Load Data from Analysis Results)

```
*deck,ElResultStrt
      subroutine ElResultStrt (Label,Comp,LabAvg,TypeData,nVal,iLoc)
c *** primary function:    (post1) Load data for later ElResultGet

c *** Notice - This file contains ANSYS Confidential information ***
c               (prolog is not confidential)

c  input arguments:
c     Label    (ch*4,sc,in)          - Result Type
c     Comp     (ch*4,sc,in)          - Result Component (8 char for ESTR)
c     LabAvg   (ch*4,sc,in)          - 'AVG ' or 'NOAV' ('AVG ' default)

c  output arguments:
```

```
c     TypeData (int,sc,out)          - Code for data type
c     nVal     (int,sc,out)          - Number of values per point
c                                      If 0, no data
c     iLoc      (int,sc,out)          - Location of Comp in values
```

2.10.4. Subroutine ElResultGet (Getting Results Values at Selected Points)

```
*deck,ElResultGet
      subroutine ElResultGet (nPoints,ebest,elcord,TypeData,iLoc,
     x                        nVal,result)
c *** primary function:    (post1) Get results at selected points

c *** Notice - This file contains ANSYS Confidential information ***
c               (prolog is not confidential)

c  input arguments:
c     nPoints  (int,sc,in)           - Number of evaluation points
c                     *** from ElInterp ***
c     ebest    (int,ar(nPoints),in)  - Element(s) containing points
c     elcord   (dp,ar(3,nPoints),in) - Element coordinates
c                     *** from ElResultStrt ***
c     TypeData (int,sc,in)           - Data type code
c     iLoc      (int,sc,in)           - Start of selected data
c     nVal     (int,sc,in)           - Number of results per point

c  output arguments:
c     Result   (dp,ar(nvar,nPoints),out)  - Array of results
```

2.10.5. Subroutine ElInterp (Finding Element Coordinates)

```
*deck,ElInterp
      subroutine ElInterp (piFEML,nPoints,xyzPoints,tolInsidein,
     x                     tolOutsidein,MoveTol,ebest,elcord)

c primary function:    Find element numbers containing xyz points
c secondary functions: Find element coordinates of these points

c object/library: upf

c *** Notice - This file contains ANSYS Confidential information ***
c     (Prolog is not CONFIDENTIAL INFORMATION)

c input arguments:
c   piFEML   (ptr,sc,in)          - If non 0, pointer of a FEM Object
c   nPoints  (int,sc,in)          - Number of points to find (do in one group)
c   xyzPoints(dp,ar(3,nPoints),in)- XYZ coordinates of each point
c   tolInsidein(dp,sc,in)         - Tolerance for point inside element
c                                     (0.0d0 defaults to 1.0d-4)
c   tolOutsidein(dp,sc,in)        - Maximum distance outside to be associated
c                                     with an element (0.0d0 defaults to 0.25)
c   MoveTol   (dp,sc,in)          - Node move tolerance (0.0d0, no move)

c output arguments:
c   ebest    (int,ar(nPoints),out)  - Best element number for each point
c   elcord   (dp,ar(3,nPoints),out) - Element coordinates of the point
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.11. Access at the Beginning and End of Various Operations

You can access the logic just before a run begins or just after a run ends, and at many other intermediate points, by using the subroutines listed below. These subroutines can perform actions such as evaluating results or performing calculations. (None of the subroutines have input or output arguments.)

Issue the **USRCAL** command (or use an equivalent menu path) to activate or deactivate these subroutines.

User Subroutine	Is Called
UAnBeg [1]	At start-up
USolBeg	Before solution
ULdBeg	Before a load step
USsBeg	Before a substep
UItBeg	Before an iteration
UItFin	After an iteration
USsFin	After a substep
ULdFin	After a load step
USolFin	After solution
UAnFin	At the end of a run

1. The UAnBeg subroutine that allows user access at the start of a run does not require activation by the **USRCAL** command; it is automatically activated when the program is started.

Subroutines USSBeg, UItBeg, UItFin and USSFin default to reading a command macro file from the current working directory whose name is subroutine.mac (that is, ussfin.mac is read by USSFin.F). No user action to relink the program is required for the command macro to be read except that the calling subroutine must be activated by the **USRCAL** command. The design of the command reading ability of these subroutines is limited to APDL parameter setting commands (***GET**, ***SET**, a = value, etc) and testing for general commands is limited. Commands which are known to work include ***DIM**, ***STATUS**. Commands which require another line (***MSG**, ***VWRITE**) are not allowed. Other commands which are known to not work are the solution loading commands (**D**, **F**, **SFE**, and so on). If these capabilities are required, the user will need to create a FORTRAN subroutine and link this subroutine into the program, as described in Understanding User Programmable Features (UPFs) (p. 127).

While parameter substitution into commands is not permitted, USSBeg, and so on were designed to be used in conjunction with dynamic tables and parameter substitution from the user subroutine. As an example, consider a table defined as d5 = f(par1), If d5 contains values of displacement as a function of PAR1, then d5 may be used as a constraint, as

```
*dim,d5,table,10,1,1,PAR1
d5(1)=0,.1,.25,

/solu
d,5,ux,%d5%
```

Modify the value of PAR1 in USSBeg.MAC and the constraint on node 5, ux can then be modified in the middle of a load step.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following is an example of a valid input that may be read by USSBeg, UItBeg, UItFin and USSFin.

```
/COM, SAMPLE ussfin.mac
a=5
b=nx(1)                          ! *get function is ok
*get,c,active,solu,Time,cpu      ! *get is ok
*dim,array,,6                    ! array parameters are ok
array(1) = 1
array(2) = 2
array(3) = 3
array(4) = 4
array(5) = 5
array(6) = 6
*vleng,3                          ! vector operations are ok
*vfun,array(4),copy,array(1)
*stat
*stat,array(1)
array(1)=
nnode = ndinqr(0,14)
*dim,array,,nnode
*vget,array(1),NODE,1,NSEL
*stat,array(1)
array(1)=
/eof

/COM, COMMANDS BELOW THIS LINE ARE KNOWN TO NOT WORK

p,1,6,2000                       ! commands DO NOT work
d,1,uy,.1
*msg,note
THIS IS A TEST MESSAGE
*vwrite,array(1)
(/ b = ,f10.4)
```

2.12. Memory-Management Subroutines

The program uses a dynamic memory manager that overlays the system *malloc* and *free* functions and provides a mechanism for accessing the memory from FORTRAN as well as C and C++. The memory manager library for Windows and Linux are:

Windows: The memory manager is in a dynamic linked library, `\Program Files|ANSYS Inc\v2020\ANSYS\bin\<platform>\ansMemManager.dll`, where <*platform*> is a directory that uniquely identifies the hardware platform version.

Linux: The memory manager is a shared library in `/ansys_inc/v202/ansys/lib/<platform>/libansMemManager.so`

You may use the system *malloc* and *free* functions or, for FORTRAN, the *allocate* system function. However, you may end up competing with the program for memory, and for large problems there may be insufficient system memory to perform the function.

Alternatively, you can use Ansys subroutines for memory management.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.12.1. Using the Memory Manager in a FORTRAN UPF

In FORTRAN, dynamic memory is done through *Cray-style* pointers, where a dynamically allocated array is defined via the construct

```
pointer (piArray,Array)
integer Array(*)
```

and memory space for the array is allocated by assigning the pointer, in this case piArray, to the allocated memory space:

```
piArray = fAnsMemAlloc (size,...)
```

or

```
piArray = fAnsMemAllocL (sizeL,...)
```

where size in an integer (4-byte) length variable, and sizeL is a long integer (8-byte) length variable.

To use the memory manager in a FORTRAN UPF, follow these steps:

1. Define the dynamically allocated arrays:

   ```
   pointer (piArray,Array), (pdArray,dArray)
   integer Array(*)
   double precision dArray(*)
   ```

2. Initialize the pointers as follows:

   ```
   piArray = PTRFTNNULL
   pdArray = PTRFTNNULL
   ```

3. Allocate space for an array or arrays, as follows:

   For integer (4-byte) numbers:

   ```
   piArray = fAnsMemAlloc(ileng,MEM_INTEGER,C16Label)
   ```

   For long integer (8-byte) numbers:

   ```
   piArray = fAnsMemAlloc(ileng,MEM_LONGINT,C16Label)
   ```

   For double-precision numbers:

   ```
   pdArray = fAnsMemAlloc(dleng,MEM_DOUBLE,C16Label)
   ```

   For complex numbers:

   ```
   pcArray = fAnsMemAlloc(cleng,MEM_COMPLEX,C16Label)
   ```

   For real numbers:

   ```
   prArray = fAnsMemAlloc(rleng,MEM_REAL,C16Label)
   ```

   Where the arguments are:

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- *x*leng is the desired size of the array (use `fAnsMemAllocL` when this size exceeds 2e31 in value)
- MEM_*xxx* is the keyword indicating the type of data
- C16Label is a character*16 name of the memory block

You must include the `ansysdef.inc` include file to get the parameter values of MEM_INTEGER, MEM_LONGINT, MEM_DOUBLE, MEM_COMPLEX, and MEM_REAL. The parameter value PTRFTNNULL is defined in impcom.inc.

Note:

If there is insufficient memory, `fAnsMemAlloc` and `fAnsMemAllocL` return "PTRFTNNULL".

4. Use the arrays.
5. If necessary, you may either shrink or grow the allocated memory space by using the reallocation routine. In this case, the original pointer and new length must be passed in as follows:

```
piArray = fAnsMemRealloc(piArray,ileng,MEM_INTEGER,C16Label)
```

or

```
piArray = fAnsMemReallocL(piArray,ilengL,MEM_INTEGER,C16Label)
```

6. Deallocate the space using the `fAnsMemFree` subroutine, as follows:

```
call fAnsMemFree (piArray)
```

The next sections provide Input and output listings for the memory management subroutines in FORTRAN.

For an example using the memory management functions, see: Function user03 (Demonstrates Using Memory) (p. 270) discussed in Defining Your Own Commands (p. 267).

2.12.1.1. Function fAnsMemAlloc (Allocating Space and Returning a Pointer)

```
*deck,fAnsMemAlloc
      function fAnsMemAlloc (iLeng, key, c16Label)

c primary function:    Get A Block of Space from mem manager and Return Pointer

c object/library:  mem

c *** Notice - This file contains ANSYS Confidential information ***

c  input arguments:
c     iLeng        (int,sc,in)       - length of the block (in data elements)
c     c16Label     (chr*16,sc,in)    - 16 character name for the Block
c     key          (int,sc,in)       - type of data for this block (see ansysdef.inc)

c  output arguments:
c     fAnsMemAlloc (PTRFTN,sc,out)   - Pointer to this data block -- needs to be
c                                      tied to a local variable in the calling
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                     routine
```

2.12.1.2. Function fAnsMemAllocL (Allocating Space and Returning a Pointer - long integer)

```
*deck,fAnsMemAllocL
      function fAnsMemAllocL (iLengL, key, c16Label)

c primary function:    Get A Block of Space from mem manager and Return Pointer

c object/library:  mem

c *** Notice - This file contains ANSYS Confidential information ***

c  input arguments:
c     iLengL        (LONG,sc,in)      - length of the block (in data elements)
c     c16Label      (chr*16,sc,in)    - 16 character name for the Block
c     key           (int,sc,in)       - type of data for this block (see ansysdef.inc)

c  output arguments:
c     fAnsMemAllocL (PTRFTN,sc,out)   - Pointer to this data block -- needs to be
c                                       tied to a local variable in the calling
c                                       routine
```

2.12.1.3. Function fAnsMemRealloc (Reallocating Space and Returning a Pointer)

```
*deck,fAnsMemRealloc
      function fAnsMemRealloc (memPtr, iLeng, key, c16Label)

c primary function:    Modify a Block of Space from mem manager and
c                      Return Pointer

c object/library:  mem

c *** Notice - This file contains ANSYS Confidential information ***

c  input arguments:
c     memPtr         (PTRFTN,sc,in)    - pointer of block being reallocated
c     iLeng          (int,sc,in)       - new length of the block (in data elements)
c     key            (int,sc,in)       - type of data needed (see ansysdef.inc)
c     c16Label       (ch*16,sc,in)     - name for this block

c  output arguments:
c     fAnsMemRealloc (PTRFTN,sc,out)   - Pointer to the new block location
```

2.12.1.4. Functional fAnsMemReallocL (Reallocating Space and Returning a Pointer - long integer)

```
*deck,fAnsMemReallocL
      function fAnsMemReallocL (memPtr, iLengL, key, c16Label)

c primary function:    Modify a Block of Space from mem manager and
c                      Return Pointer
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c object/library:  mem

c *** Notice - This file contains ANSYS Confidential information ***

c  input arguments:
c     memPtr          (PTRFTN,sc,in)    - pointer of block being reallocated
c     iLengL          (LONG,sc,in)      - new length of the block (in data elements)
c     key             (int,sc,in)       - type of data needed (see ansysdef.inc)
c     c16Label        (ch*16,sc,in)     - name for this block

c  output arguments:
c     fAnsMemReallocL (PTRFTN,sc,out)   - Pointer to the new block location
```

2.12.1.5. Subroutine fAnsMemFree (Deallocating Space)

```
*deck,fAnsMemFree
      subroutine fAnsMemFree (memPtr)

c primary function:   Free a Data Block, given a pointer

c object/library:  mem

c *** Notice - This file contains ANSYS Confidential information ***

c  input arguments:
c     ptr   (PTRFTN,sc,inout)  - pointer for this block

c  output arguments:
c     ptr   (PTRFTN,sc,inout)  - pointer will be set to PTRFTNNULL
```

2.12.2. Using the Memory Manager in a C or C++ UPF

To use the memory manager In C or C++, follow these steps:

1. Define the dynamically allocated arrays:

   ```
   int *iArray;
   double *dArray;
   ```

2. Initialize the pointers as follows:

   ```
   iArray = (int*) NULL;
   dArray = (double*) NULL;
   ```

3. Allocate space for an array or arrays, as follows:

   For integer (4-byte) numbers:

   ```
   numBytes = leng*sizeof(int);
   iArray = cAnsMemAlloc(numBytes,0,__FILE__,__LINE__);
   ```

   For double-precision numbers:

   ```
   numBytes = leng*sizeof(double);
   dArray = cAnsMemAlloc(numBytes,0,__FILE__,__LINE__);
   ```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Where the arguments are:

- numBytes is the desired amount of memory to allocate (NOTE: this length can be a 64-bit integer)
- The __FILE__ and __LINE__ arguments are used to create a string for naming the memory block

> **Note:**
>
> If there is insufficient memory, cAnsMemAlloc will return a NULL pointer.

4. Use the arrays.
5. If necessary, you may either shrink or grow the allocated memory space by using the reallocation routine. In this case, the original pointer and new length must be passed in as follows:

```
iArray = cAnsMemRealloc(iArray,newNumBytes,0,__FILE__,__LINE__);
```

6. Deallocate the space using the cAansMemFree subroutine, as follows:

```
call cAnsMemFree (iArray,__FILE__,__LINE__);
```

> **Note:**
>
> the pointer passed into cAnsMemFree is untouched upon leaving this routine.

The next sections provide Input and output listings for the memory management subroutines in C or C++.

2.12.2.1. Function cAnsMemAlloc (Allocating Space and Returning a Pointer)

```
void *cAnsMemAlloc(size_t iLen, INT key, CHAR *cName, INT cLineNum)
/*
 * Allocate a block of memory from the memory manager and return pointer
 *
 * PARAMETER LIST:
 *  iLen     - length of block to allocate (in bytes)
 *  key      - bit patterned descriptive key to be passed to lower level manager
 *             =0, get memory any way possible
 *             &MEM_INITIAL, get memory only from initial heap block
 *             &MEM_GROWTH, get memory only outside of initial heap block
 *
 *             also stores bit patterns (in bits 17-23) representing various
 *              allocation groups (solvers, etc)
 *  cName    - name of calling routine (should be gotten from __FILE__)
 *  cLineNum - line number of calling routines (gotten from __LINE__)
 *
 * RETURN VALUE:
 *  a pointer to the memory space
 */
```

2.12.2.2. Function cAnsMemRealloc (Reallocating Space and Returning a Pointer)

```
void *cAnsMemRealloc(void *memPtr, size_t iLen, INT key, CHAR *cName, INT cLineNum)
/*
 * Modify a block of memory from the memory manager and return pointer
 *
 * PARAMETER LIST:
 *  memPtr   - pointer to the memory block being reallocated
 *  iLen     - length of block to reallocate (in bytes)
 *  key      - bit patterned descriptive key to be passed to lower level manager
 *             =0, get memory any way possible
 *             &MEM_INITIAL, get memory only from initial heap block
 *             &MEM_GROWTH, get memory only outside of initial heap block
 *
 *             also stores bit patterns (in bits 17-23) representing various
 *              allocation groups (solvers, etc)
 *  cName    - name of calling routine (should be gotten from __FILE__)
 *  cLineNum - line number of calling routines (gotten from __LINE__)
 *
 * RETURN VALUE:
 *  a pointer to the memory space
 */
```

2.12.2.3. Subroutine cAnsMemFree (Deallocating Space)

```
void cAnsMemFree(void *memPtr, CHAR *cName, INT cLineNum)
/*
 * Free a block of memory, given a pointer
 *
 * PARAMETER LIST:
 *  memPtr   - pointer to the memory block being deallocated
 *  cName    - name of calling routine (should be gotten from __FILE__)
 *  cLineNum - line number of calling routines (gotten from __LINE__)
 *
 * RETURN VALUE:
 *  a pointer to the memory space (will be set to NULL)
 */
```

2.13. Parameter-Processing Subroutines

The product distribution medium contains three subroutines that you can use for parameter processing: `pardim`, `parevl`, and `pardef`.

2.13.1. Subroutine pardim (Creating a Dimensioned Parameter)

```
*deck,pardim
      subroutine pardim (cName,labl4,nDim,nxyz,cLabels)
c *** primary function:    create a dimensioned parameter

c       *dim,parm32,type,d1,d2,d3,cName1,cName2,cName3
c       *dim,parm32,type,d1,cName1
c       *dim,parm32,type,d1,d2,d3,d4,d5,cName1,cName2,cName3,cName4,cName5

c *** Notice - This file contains ANSYS Confidential information ***

c  input arguments:
c     cName    (chr*32,sc,in)    - the name of the parameter to create
c     labl4    (chr*4,sc,in)     - 'TABL' or 'ARRA' or 'CHAR' or 'STRI'
c     nDim     (int,sc,in)       - Dimension of array
c     nxyz     (int,ar(nDim),in) - the dimensions of the array
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c     cLabels  (chr*32,ar(nDim),in) - Names for the directions in table

c  output arguments:  none
```

2.13.2. Subroutine parevl (Finding and Evaluating a Parameter)

```
*deck,parevl
      subroutine parevl (ParName,nDim,subc,lvl,dpValue,chValue,kerr)
c *** primary function:    find and evaluate a parameter

c *** Notice - This file contains ANSYS Confidential information ***

c  input arguments:
c     ParName  (chr*(PARMSIZE),sc,in) - the name of the parameter
c                                       (must be upper case, left justified)
c     nDim     (int,sc,in)            - the number of subscripts (0,scaler)
c     subc     (dp,ar(*),in)          - values for the subscripts (if any)
c     lvl      (int,sc,in)            - 0,1  no error output  2, report error
c                                       -1, set kerr flag with no anserr call

c  output arguments:
c     dpValue  (dp,sc,out)            - the value of the parameter (may be a
c                                       packed character*8
c     chValue  (chr*(STRING_MAX_LENG),sc,out) - character output
c     kerr     (int,sc,out)           - error flag  (0,ok  -1,output is packed
c                                       0=ok,  1=error,  2=error but TINY is used
c                                        -2, output is string in chValue
```

2.13.3. Subroutine pardef (Adding a Parameter)

```
*deck,pardef
      subroutine pardef (cNameIn,ctype,nval,subc,valuein,kerr,string)
c *** primary function:    add a parameter to parameter list
c
c *** Notice - This file contains ANSYS Confidential information ***
c
c  input arguments:
c     cNameIn (chr*(PARMSIZE),sc,in) - name of parameter
c                              cNameIn is a character variable that
c                              contains the name of the parameter that
c                              is to be defined. (Length = PARMSIZE characters)
c
c     ctype  (int,sc,in)   - 0, dp   1,character  2,string
c                              ctype is an integer key which describes
c                              the type of data that the parameter data
c                              holds.  This would also indicate the
c                              contents of "value" (arg 5).
c                              0=double precision data
c                              1=character data packed in value
c                              2=character data in string
c
c     nval   (int,sc,in)   - number of subscripts
c                              nval is the number of subscripts that the
c                              "cNameIn" (arg 1) contains.
c                              1=single dimensioned variable (ex. x(10))
c                              2=double dimensioned variable (ex. y(10,3))
c                              3=triple dimensioned variable (ex. z(10,3,2))
c                              -1=delete this parameter from the internal
c                              tables.
c
c     subc (dp,ar(*),in)   - values of subscripts
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                               subc is a double precision vector that
c                               contains the subscripts of "cNameIn" (arg 1).
c                               There should be enough values defined to
c                               match "nval" (arg 3).  For example if "x"
c                               was dimensioned as "x(10,3,2)" and you wanted
c                               to set "x(5,1,1)=123.0", then "nval" (arg 3)
c                               should be set to 3, and "subc" should be set
c                               to 5.0, 1.0, 1.0, and "value" (arg 5) should
c                               be 123.0.  Another example is if "y" was
c                               dimensioned to as "y(20,20)" and you were
c                               setting "y(5,8)=987", then "nval" (arg 3) should
c                               be set to 2 and "subc" should be set to 5.0,
c                               8.0, 0.0, and "value" (arg 5) should be 987.0.
c
c                               Remember subroutine "pardef" is only storing
c                               a data value of "cNameIn" or "cNameIn(x,y,z)". The
c                               proper dimensions were set by a "*dim" command.
c
c                               Please note that although the values of "subc"
c                               should be double precision, subroutine "pardef"
c                               uses the intrinsic "nint" (nearest integer)
c                               function to get and use the integer equivalent.
c
c                              You should also note the "nval" (arg 3) and
c                             "subc" (arg 4) must fall within the range that was
c                               set with a "*dim" or "*set" command or an error
c                               will occur.
c
c     valuein(dp,sc,in)    - the value for this parameter
c                               (should be a packed character*8 if
c                               ctype=1. To pack a char into a dp
c                               variable use "call chtodp(ch8,dp)".
c                               To unpack a dp variable into a char
c                               use "call dptoch(dp,ch8)" )
c                               Value is the data value that is to be stored for
c                               "cNameIn" (arg 1). If "ctype=1" (arg 2) then this
c                               value would be a "packed character" data from the
c                               "chtodp" Ansys function.
c
c  output arguments:
c     kerr   (int,sc,out)  - error flag (0=ok, 1=error)
c                               kerr is an integer error flag that is
c                               returned to the calling subroutine.  Any
c                               non zero number would indicate an error
c                               was detected in subroutine "pardef"
c
c *** mpg pardef < parstore pardim ntableget rdsset<rdmac<rdcmd: define param
c
```

2.14. Other Useful Functions

The program has several miscellaneous functions that you may find useful for working with UPFs:

- The `erhandler` subroutine displays output messages (notes, warnings, and errors).
- The `RunCommand` function lets you issue a command from within a user subroutine.
- The `GetStackDisp` subroutine retrieves current displacement values.
- The **/UNDO** command writes an "undo" file at critical points as a user subroutine executes.
- The **/HOLD** command allows you to synchronize multiple tasks.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For further descriptions of `erhandler`, see Subroutines for Your Convenience (p. 351). For details about the `GetStackDisp` function, see Function GetStackDisp (Getting Current Displacement Values) (p. 275).

2.14.1. Using Function RunCommand

This function enables you to issue a command from within a user subroutine. Inputs and outputs for `RunCommand` are as follows:

```
*deck,RunCommand
      function RunCommand (nChar,command)

c primary function:    Execute an ansys command

c object/library: upf

c *** Notice - This file contains ANSYS Confidential information ***

c  input arguments:
c     nChar    (int,sc,in)           - Length of the command string (8 min)
c     command  (ch*(nChar),sc,in)    - A character string containing a
c                                              valid ANSYS command

c  output arguments:
c     RunCommand   (int,sc,out)      - An internally defined value, ignore
```

2.14.2. Using the /UNDO Command

The "undo" file you create by issuing the **/UNDO** command is similar to the `File.DB` file created when you issue the **SAVE** command. The **/UNDO** command format is:

```
/UNDO,Action
```

`Action`

ON, to write the undo file

OFF, to prevent the undo file from being written

PROMPT, to have the program ask permission before writing the file

STATUS, to restore the file as it existed after executing the last command issued before the **/UNDO** command.

2.14.3. Using the /HOLD command

Issue the **/HOLD** command to synchronize tasks. The program can synchronize tasks at the end of each results file set.

```
/HOLD,Filename,TimeInterval,Timeout
```

`Filename`

The eight-character name of a message file. If the named file exists, the program reads a command from the file and then deletes the file.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

TimeInterval

The length of time, in seconds, that the program waits before trying to read the message file again.

Timeout

The maximum length of time, in seconds, that the program can wait between attempts to read the file.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 3: Accessing the Mechanical APDL Database

This chapter describes how you can retrieve information in the Mechanical APDL database (or store information in the database) by linking subroutines you create into the Mechanical APDL program.

You can use the database access routines with any of the user-programmable features. For example, you can create your own Mechanical APDL commands and use them to execute database access routines (or have a database access routine call a user-defined command).

Inputs and Outputs for Database Access Routines

The descriptions of the database access routines or functions within this chapter describe both the input arguments and output arguments. Argument information includes the argument's type, size and intent.

- Argument *type* is one of the following:

  int - integer (4-byte)
  long - integer (8-byte)
  dp - double precision
  log - logical
  chr - character
  comp - double precision complex

- Argument *size* is one of the following:

  sc - scalar variable
  ar(*n*) - array variable of length *n*
  func - functional return value

- Argument *intent* is one of the following:

  in - input argument
  out - output argument
  inout - both an input and an output argument

Types of Database Access Routines

The rest of this chapter describes the functions and subroutines available for accessing information in the Mechanical APDL database. The function and subroutine descriptions are grouped into the following sections.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.1. Routines for Selecting and Retrieving Nodes and Elements

3.1.1. Function ndnext (Getting the Next Node Number)

```
*deck,ndnext
      function ndnext (next)
c *** primary function:    get the number of the next selected node

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        next     (int,sc,in)       - the last node number used
c                                     = 0 - use for initial value

c     output arguments:
c        ndnext   (int,func,out)    - the next selected node number
c                                     = 0 - no more nodes
```

3.1.2. Function ndprev (Getting the Number of the Previous Selected Node)

```
*deck,ndprev
      function ndprev (next)
c *** primary function:    get the number of the previous selected node
c
c *** Notice - This file contains ANSYS Confidential information ***
c
c     typ=int,dp,log,chr,dcp   siz=sc,ar(n),func    intent=in,out,inout
c
c     input arguments:
c     variable (typ,siz,intent)    description
c        next     (int,sc,in)       - the next node number used
c                                     = 0 - use for initial value
c
c     output arguments:
c        ndprev   (int,func,out)    - the previous selected node number
c                                     = 0 - no more nodes
c
```

3.1.3. Function ndnxdf (Getting the Number of the Next Defined Node)

```
*deck,ndnxdf
      function ndnxdf (next)
c *** primary function:    get the number of the next defined node
c
c *** Notice - This file contains ANSYS Confidential information ***
c
c     input arguments:
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c        next     (int,sc,in)       - the last node number used
c                                     = 0 - use for initial value
c
c     output arguments:
c        ndnxdf   (int,func,out)    - the next defined node number
c                                     = 0 - no more nodes
```

3.1.4. Function ndsel (Selecting, Unselecting, Deleting, or Inverting a Node)

```
*deck,ndsel
      subroutine ndsel (ndmi,ksel)
c *** primary function:    to select, unselect, delete, or invert a node.
c *** secondary functions: none.

c *** Notice - This file contains ANSYS Confidential information ***

c     typ=int,dp,log,chr,dcp   siz=sc,ar(n),func    intent=in,out,inout

c     input arguments:
c        variable (typ,siz,intent)    description
c        ndmi     (int,sc,in)       - node number
c                                     = 0 - all nodes
c                                     < 0 - do not delete CPs and CEQNs
c                                           (merge/offset/compress)
c        ksel     (int,sc,in)       - type of operation to be performed.
c                                     ksel = 0 - delete node.
c                                          = 1 - select node.
c                                          =-1 - unselect node.
c                                          = 2 - invert select status of node.

c     output arguments:
c        none.
```

3.1.5. Function elnext (Getting the Number of the Next Element)

```
*deck,elnext
      function elnext (next)
c *** primary function:    get the number of the next selected element

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        next     (int,sc,in)       - the last element number used
c                                     = 0 - use for initial value

c     output arguments:
c        elnext   (int,func,out)    - the next selected element
c                                     = 0 - no more elements
```

3.1.6. Function elprev (Getting the Number of the Previous Selected Element)

```
*deck,elprev
      function elprev (prev)
c *** primary function:    get the number of the previous selected element
c
```

```
c *** Notice - This file contains ANSYS Confidential information ***
c
c     typ=int,dp,log,chr,dcp   siz=sc,ar(n),func    intent=in,out,inout
c
c     input arguments:
c     variable (typ,siz,intent)     description
c        prev     (int,sc,in)       - the last element used
c                                     = 0 - use for initial value
c
c     output arguments:
c        elprev   (int,func,out)    - the previous selected element
c                                     = 0 - no more elements
c
```

3.1.7. Function elnxdf (Getting the Number of the Next Defined Element)

```
*deck,elnxdf
      function elnxdf (next)
c *** primary function:    get the number of the next defined element
c
c *** Notice - This file contains ANSYS Confidential information ***
c
c     input arguments:
c        next     (int,sc,in)       - the last element number used
c                                     = 0 - use for initial value
c
c     output arguments:
c        elnxdf   (int,func,out)    - the next defined element
c                                     = 0 - no more elements
```

3.1.8. Subroutine elsel (Selecting, Unselecting, Deleting, or Inverting an Element)

```
*deck,elsel
      subroutine elsel (ielei,ksel)
c *** primary function:    to select, unselect, delete, or invert an element.

c *** Notice - This file contains ANSYS Confidential information ***
c     input arguments:
c        ielei    (int,sc,in)       - element number
c                                     = 0 - all elements
c        ksel     (int,sc,in)       - type of operation to be performed.
c                                     = 0 - delete element.
c                                     = 1 - select element.
c                                     =-1 - unselect element.
c                                     = 2 - invert select status for element
c     output arguments:  none
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.2. Node Information Routines

3.2.1. Function ndinqr (Getting Information About a Node)

The primary function of `ndinqr` is getting information about a node. This function also sets the current node pointer to this node.

Note:

Some of the database commands in the input file shown below are in the common block `ansysdef.inc`, which must be included in the subroutine.

```
*deck,ndinqr
      function ndinqr (node,key)
c *** primary function:    get information about a node.
c *** secondary functions: set current node pointer to this node.

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        node     (int,sc,in)       - node number
c                                      Should be 0 for key=11, DB_NUMDEFINED,
c                                      DB_NUMSELECTED, DB_MAXDEFINED, and
c                                      DB_MAXRECLENG
c        key      (int,sc,in)       - key as to information needed about
c                                     the node.
c                 = DB_SELECTED    - return select status:
c                     ndinqr  =  0 - node is undefined.
c                             = -1 - node is unselected.
c                             =  1 - node is selected.
c                 = DB_NUMDEFINED  - return number of defined nodes
c                 = DB_NUMSELECTED - return number of selected nodes
c                 = DB_MAXDEFINED  - return highest node number defined
c                 = DB_MAXRECLENG  - return maximum record length (dp words)
c                 =   2, return length (dp words)
c                 =   3,
c                 =   4, pointer to first data word
c                 =  11, return void percent (integer)
c                 =  17, pointer to start of index
c                 = 117, return the maximum number of DP contact data stored for any node
c                 =  -1,
c                 =  -2, superelement flag
c                 =  -3, master dof bit pattern
c                 =  -4, active dof bit pattern
c                 =  -5, solid model attachment
c                 =  -6, pack nodal line parametric value
c                 =  -7, constraint bit pattern
c                 =  -8, force bit pattern
c                 =  -9, body force bit pattern
c                 = -10, internal node flag
c                 = -11, orientation node flag =1 is =0 isnot
c                 = -11, contact node flag <0
c                 = -12, constraint bit pattern (for DSYM)
c                 = -13, if dof constraint written to file.k (for LSDYNA only)
c                 = -14, nodal coordinate system number (set by NROTATE)
c                 =-101, pointer to node data record
c                 =-102, pointer to angle record
c                 =-103,
c                 =-104, pointer to attached couplings
c                 =-105, pointer to attacted constraint equations
c                 =-106, pointer to nodal stresses
c                 =-107, pointer to specified disp'S
c                 =-108, pointer to specified forces
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                =-109, pointer to x/y/z record
c                =-110,
c                =-111,
c                =-112, pointer to nodal temperatures
c                =-113, pointer to nodal heat generations
c                =-114,
c                =-115, pointer to calculated displacements
c                =-116,

c     output arguments:
c        ndinqr   (int,func,out)   - the returned value of ndinqr is based on
c                                       setting of key.
```

3.2.2. Function getnod (Getting a Nodal Point)

```
*deck,getnod
      subroutine getnod (node,v,kerr,kcrot)
c *** primary function:    get a nodal point

c *** Notice - This file contains ANSYS Confidential information ***

c  input arguments:
c     node     (int,sc,in)       - node number
c     kerr     (int,sc,inout)    - message flag
c                                  = 0 - print no message if node is unselected
c                                        or undefined
c                                  = 1 - print message if node is undefined
c                                  = 2 - print message if node is undefined
c                                        or unselected
c     kcrot    (int,sc,in)       - output coordinates in this coordinate system.
c                                    if kcrot is negative, output theta and
c                                    phi coordinates in radians

c  output arguments:
c     v        (dp,ar(6),out)    - Coordinates (first 3 values) and rotation
c                                    angles (last 3 values)
c     kerr     (int,sc,inout)    - select status
c                                  = 0 - node is selected
c                                  = 1 - node is not defined
c                                  =-1 - node is unselected
```

3.2.3. Function putnod (Storing a Node)

```
*deck,putnod
      subroutine putnod (node,vctn,kcrot)
c *** primary function:    store a node
c *** secondary functions: display node if in immediate mode.

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        node     (int,sc,in)       - node number
c        vctn     (dp,ar(6),in)     - array of 3 nodal coordinates and
c                                              3 nodal rotation angles.
c        kcrot    (int,sc,in)       - local coordinate system in which the nodal
c                                       coordinates and angles are defined

c     output arguments:  none.
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.2.4. Function ndgall (Getting the XYZ/Rotation Coordinates Vector for a Node)

```
*deck,ndgall
      function ndgall (node,xyz)
c *** primary function:    get x,y,z,rotx,roty,rotz vector for a node.

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        node     (int,sc,in)       - node number for operation.

c     output arguments:
c        ndgall   (int,sc,out)      - status of node.
c                                        0=node is undefined.
c                                       -1=node is unselected.
c                                        1=node is selected.
c        xyz      (dp,ar(6),out)    - vector containing x,y,z,rotx,roty,rotz
```

3.2.5. Subroutine ndspgt (Getting the Nodal Solution for a Node of an Element)

```
*deck,ndspgt
      subroutine ndspgt (node,dofs,ndof,nrot,xyzang,nuvect,unode)
c *** primary function:  get the nodal solution for a node of an element

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        node     (int,sc,in)    - The node number
c        dofs     (int,ar(DOFBITLENG),in) - The dofs to retrieve for the node.
c                                    dof = degree of freedom
c                                    The dofs array should be zeroed out,
c                                    except for the needed parts.
c                                    dofs is a bit pattern with true bits
c                                    representing the GLOBAL Dof set desired.
c                                    That is, dofs(1) is used for UX to SP06,
c                                    and dofs(2) is used for TBOT to TTOP.
c                                    See ECHPRM for details.  For example,
c                                          dofs(1) = UX + TEMP
c                                          dofs(2) = TE3
c                                    TTOP is a special case.  If you want
c                                    TTOP alone, use:
c                                          dofs(2) = ibset(0,TTOP)
c                                    If TBOT and TTOP are desired, you must use:
c                                          dofs(2) = TBOT
c                                          dofs(2) = ibset(dofs(2),TTOP)
c        ndof     (int,sc,in)    - The number of node dofs (1, 2 or 3).
c        nrot     (int,sc,in)    - Key to rotate dofs from nodal to global
c                                    coordinate systems.
c                                    if 0, none. if 2, 2-d. if 3, 3-d
c                                    if > 0, dof set must include and only
c                                    include all terms of the vector (e.g.
c                                    UX,UY,UZ, or AX,AY,AZ).
c        xyzang   (dp,ar(6),in)  - The xyz virgin node coordinates
c                                     (including angles).  Not used if
c                                     nrot = 0 or ndof < 2.
c        nuvect   (int,sc,in)    - Number of vectors to retrieve.  Can vary
c                                    between 1 and 5.  Normally 1 is what is
c                                    wanted.  Other vectors include previous
c                                    values and/or velocities.  See elucom for
c                                    all possibilites.  Contents are analysis
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                  type dependent.

c     output arguments:
c        unode    (dp,ar(ndof,nuvect),out) - Element nodal solution vectors in
c                                             the global coordinate system.
```

3.3. Element Attribute Routines

3.3.1. Function elmiqr (Getting Information About an Element)

```
*deck,elmiqr
      function elmiqr (ielem,key)
c *** primary function:    get information about an element.
c *** secondary functions: set current element pointer to this element.

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem    (int,sc,in)       - element number
c                                     should be zero for key=11, DB_NUMDEFINED,
c                                       DB_NUMSELECTED, DB_MAXDEFINED, DB_MAXRECLENG,
c                                       or 199
c        key      (int,sc,in)       - information flag.
c                 = DB_SELECTED    - return select status:               (1)
c                      elmiqr = 0 - element is undefined.
c                               =-1 - element is unselected.
c                               = 1 - element is selected.
c                 = DB_NUMDEFINED  - return number of defined elements    (12)
c                 = DB_NUMSELECTED - return number of selected elements   (13)
c                 = DB_MAXDEFINED  - return maximum element number used   (14)
c                 = DB_MAXRECLENG  - return maximum record length         (15)
c                                    (int words)
c                 = 2 - return length (int words)
c                 = 3 - return layer number
c                       (for cross reference files return number of entities)
c                 = 4 - return address of first data word
c                 = 5 - return length (in record type units)
c                 = 6 - return compressed record number.
c                 = 11 - return void percent (integer)
c                 = 16 - return location of next record
c                        (this increments the next record count)
c                 = 17 - pointer to start of index
c                 = 18 - return type of file.
c                     elmiqr = 0 - integer
c                              = 1 - double precision
c                              = 2 - real
c                              = 3 - complex
c                              = 4 - character*8
c                              = 7 - index
c                 = 19 - return virtual type of file.
c                     elmiqr = 0 - fixed length (4.4 form)
c                              = 1 - indexed variable length (layer data)
c                              = 2 - xref data tables
c                              = 3 - bitmap data (for 32 data item packed records)
c                              = 4 - data tables (three dimensional arrays)
c                 = 111 - return the maximum number of nodes stored for any element
c                 = 123 - return the maximum number of DP contact data stored for any element
c                 = -1 - material number           ( = -EL_MAT)
c                   -2 - element type              ( = -EL_TYPE)
c                   -3 - real constant number      ( = -EL_REAL)
c                   -4 - element section ID number ( = -EL_SECT)
c                   -5 - coordinate system number  ( = -EL_CSYS)
c                         (see elmcmx for rest)
c                 =-101 - pointer to element integers etc.
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                           (see elmcmx with elmilg and 1 instead of -101)
c

c     output arguments:
c        elmiqr  (int,sc,out)  - the returned value of elmiqr is based on
c                                  setting of key.
c
```

3.3.2. Function elmget (Getting an Element's Attributes and Nodes)

```
*deck,elmget
      function elmget (ielem,elmdat,nodes)
c *** primary function:    get element attributes and nodes.

c *** Notice - This file contains ANSYS Confidential information ***
c     input arguments:
c        ielem      (int,sc,in)      - element number
c     output arguments:
c        elmget     (int,func,out)    -   status of element.
c                                     elmget = 0 - element undefined
c                                            < 0 - number of nodes on unselected
c                                                  element
c                                            > 0 - number of nodes on selected element
c        elmdat     (int,ar(*),out)    -   element attributes.
c                          elmdat(EL_MAT)   - material number
c                          elmdat(EL_TYPE)  - element type
c                          elmdat(EL_REAL)  - real constant number
c                          elmdat(EL_SECT)  - section number
c                          elmdat(EL_CSYS)  - coordinate system number
c                          elmdat(EL_DEAD)  - death flag (bit 0)
c                                             if clear - alive
c                                             if set   - dead
c                          elmdat(EL_SOLID) - solid model reference
c                          elmdat(EL_SHAPE) - 100*shape + specific shape
c                          elmdat(EL_SHAPE) - 100*shape + specific shape
c                          elmdat(EL_OBJOPTIONS)  - reserved
c                          elmdat(EL_PEXCLUDE) - p element include flag (bit 0)
c                                          if clear - include
c                         (element may need to have its p-level increased)
c                                          if set   - exclude
c                         (element does not need to have its p-level increased)
c                         EL_PEXCLUDE is also used for the LSDYNA part number
c        nodes     (int,ar,out)      -   node numbers for element
```

3.3.3. Subroutine elmput (Storing an Element)

```
*deck,elmput
      subroutine elmput (ielem,elmdat,nnod,nodes)
c *** primary function:    store element attributes and node numbers.
c *** secondary functions: set current element pointer to this element.

c *** Notice - This file contains ANSYS Confidential information ***

c *** NOTICE - The user is also responsible for defining the centroid for the
c              element using the elmpct subroutine.  Calling the elmput
c              subroutine will NULL the element centroid previously defined.

c     input arguments:
c        ielem    (int,sc,in)       - element number
c        elmdat   (int,ar(EL_DIM),in) - element attributes.
c                            elmdat(EL_MAT)  - material number
c                                  (EL_TYPE)  - element type
c                                  (EL_REAL)  - real constant number
```

```
c                                    (EL_SECT)  - section number
c                                    (EL_CSYS)  - coordinate system number
c                                    (EL_DEAD)  - death flag (bit 0)
c                                            if clear - alive
c                                            if set   - dead
c                                    (EL_SOLID) - solid model reference
c                                    (EL_SHAPE) - 100*shape + specific shape
c                                    (EL_OBJOPTIONS)  - reserved
c                                    (EL_PEXCLUDE) - p element include flag
c                                            (bit 0)
c                                            if clear - include
c                                            if set   - exclude
c                                            For LSDYNA, it means part ID
c                                            in regular ANSYS, it is never part ID
c        nnod     (int,sc,in)       - number of nodes for this element.
c        nodes    (int,ar(*),in)    - node numbers for this element.

c     output arguments:  none.
```

3.3.4. Function etyiqr (Getting a Data Item About an Element Type)

```
*deck,etyiqr
      function etyiqr (itype,key)
c *** primary function:    get information about an element type.

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        itype    (int,sc,in)       - element type number
c                                        Should be 0 for key=11, DB_NUMDEFINED,
c                                        DB_NUMSELECTED, DB_MAXDEFINED, and
c                                        DB_MAXRECLENG
c        key      (int,sc,in)       - information flag.
c                 = DB_SELECTED    - return select status:
c                      etyiqr = 0 - element type is undefined.
c                             =-1 - element type is unselected.
c                             = 1 - element type is selected.
c                 = DB_NUMDEFINED  - return number of defined element types
c                 = DB_NUMSELECTED - return number of selected element types
c                 = DB_MAXDEFINED  - return highest element type number defined
c                 = DB_MAXRECLENG  - return maximum record length (int words)
c                = -n, return element characteristic n from etycom for element
c                       type itype.
c                      n is correlated to the parameter names in echprm.
c                      see elccmt for definitions of element characteristics.
c                      note- this will not overwrite the current setting of
c                       etycom.

c     output arguments:
c        etyiqr   (int,func,out)    - the returned value of etyiqr is based on
c                                        setting of key.
```

3.3.5. Function etyget (Getting Information About an Element Type)

```
*deck,etyget
      function etyget (itype,ielx)
c *** primary function:    get element type data.

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c        itype    (int,sc,in)       - element type number

c     output arguments:
c        etyget   (int,func,out)    - status of element type.
c                                     = 0 - element type is undefined.
c                                     < 0 - number of data items on unselected
c                                           element type.
c                                     > 0 - number of data items on selected
c                                           element type.
c        ielx     (int,ar(*),out)   - element type data. see elccmt for
c                                     description of data.

c *** mpg etyget<el117,edgcntf1,edgcntsz,edgrde,edgrecc,edgmul:get elem type
```

3.3.6. Subroutine etyput (Storing Element Type Data)

```
*deck,etyput
      subroutine etyput (itype,n,ielx)
c *** primary function:    store element type data.

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        itype    (int,sc,in)       - element type number for operation.
c        n        (int,sc,in)       - length of data vector to store.
c        ielx     (int,ar(*),in)    - element type data.  see elccmt for
c                                     description.
c     output arguments:  none
```

3.3.7. Subroutine echrtr (Getting Information About Element Characteristics)

```
*deck,echrtr
      subroutine echrtr (iott,elcdn,ielc,kerr)
c   primary function: collect all element characteristics based on
c                                                    ityp,jtyp, and keyopts

c *** Notice - This file contains ANSYS Confidential information ***
c
c     typ=int,dp,log,chr,dcp   siz=sc,ar(n),func    intent=in,out,inout
c
c  input arguments:
c     variable (typ,siz,intent)      description
c     iott     (int,sc,in)            - printout file
c     ielc     (int,ar(IELCSZ),inout) - input element characteristics
c                                       in positions 1 to 20.
c                                       (itype, jstif, keyopts, etc.)
c
c  output arguments:
c     elcdn    (chr,sc,out)           - element descriptive name as character
c                                         string
c     ielc     (int,ar(IELCSZ),inout) - input element characteristics
c                                         in positions 21 to 150.
c                                         (kdim, ishap, idegen, etc.)
c                                         see elccmt for a full list
c     kerr     (int,sc,out)           - error flag
c                                       = 0 - no errors
c                                       = 1 - errors

c
```

3.3.8. Subroutine etysel (Selecting, Unselecting, Deleting, or Inverting an Element Type)

```
*deck,etysel
      subroutine etysel (itypi,ksel)
c *** primary function:    to select, unselect, delete, or invert an
c                          element type.
c *** secondary functions: none.
c
c *** Notice - This file contains ANSYS Confidential information ***
c
c     typ=int,dp,log,chr,dcp   siz=sc,ar(n),func    intent=in,out,inout
c
c     input arguments:
c        variable (typ,siz,intent)    description
c        itypi    (int,sc,in)       - element type number
c                                       = 0 - all element types
c        ksel     (int,sc,in)       - type of operation to be performed.
c                                       = 0 - delete element type.
c                                       = 1 - select element type.
c                                       =-1 - unselect element type.
c                                       = 2 - invert element type.
c
c     output arguments:
c        none.
c
```

3.3.9. Function mpinqr (Getting Information About a Material Property)

```
*deck,mpinqr
      function mpinqr (mat,iprop,key)
c *** primary function:    get information about a material property.

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        mat      (int,sc,in)       - material number
c                                        should be 0 for key=11,
c                                        DB_NUMDEFINED(12),
c                                        DB_MAXDEFINED(14), and
c                                        DB_MAXRECLENG(15)

c        iprop    (int,sc,in)       - property reference number:
c         if iprop = 0, test for existence of any material property with this
c                  material number (with key = DB_SELECTED(1))
c
c        ---- MP command labels --------
c        EX  = 1, EY  = 2, EZ  = 3, NUXY= 4, NUYZ= 5, NUXZ= 6, GXY = 7, GYZ = 8
c        GXZ = 9, ALPX=10, ALPY=11, ALPZ=12, DENS=13, MU  =14, DAMP=15, KXX =16
c        KYY =17, KZZ =18, RSVX=19, RSVY=20, RSVZ=21, C   =22, HF  =23, VISC=24
c        EMIS=25, ENTH=26, LSST=27, PRXY=28, PRYZ=29, PRXZ=30, MURX=31, MURY=32
c        MURZ=33, PERX=34, PERY=35, PERZ=36, MGXX=37, MGYY=38, MGZZ=39, EGXX=40
c        EGYY=41, EGZZ=42, SBKX=43, SBKY=44, SBKZ=45, SONC=46, DMPS=47, ELIM=48
c        USR1=49, USR2=50, USR3=51, USR4=51, FLUI=53, ORTH=54, CABL=55, RIGI=56
c        HGLS=57, BVIS=58, QRAT=59, REFT=60, CTEX=61, CTEY=62, CTEZ=63, THSX=64,
c        THSY=65, THSZ=66, DMPR=67, LSSM=68, BETD=69, ALPD=70, RH  =71, DXX =72,
c        DYY =73, DZZ =74, BETX=75, BETY=76, BETZ=77, CSAT=78, CREF=79, CVH =80,
c        UMID=81, UVID=82
c
c               (see mpinit for uncommented code and for TB command information)

c        key      (int,sc,in)       - key as to the information needed
c                                      about material property.
c             = DB_SELECTED(1)- return select status:
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                        mpinqr = 0 - material prop is undefined.
c                               = 1 - material prop is selected.
c            = DB_NUMDEFINED(12) - number of defined material properties
c            = DB_MAXDEFINED(14) - highest material property number defined
c            = DB_MAXRECLENG(15) - maximum record length (dp words)
c            =  2 - return length (dp words)
c            =  3 - return number of temp. values
c            = 11 - return void percent (integer)

c     output arguments:
c        mpinqr   (int,func,out)    - returned value of mpinqr is based on
c                                     setting of key.
```

3.3.10. Function mpget (Getting a Material Property Table)

```
*deck,mpget
      function mpget (mat,iprop,temp,prop)
c *** primary function:    get a material property table.

c *** Notice - This file contains ANSYS Confidential information ***

c     typ=int,dp,log,chr,dcp   siz=sc,ar(n),func    intent=in,out,inout

c     input arguments:
c        variable (typ,siz,intent)    description
c        mat      (int,sc,in)       - material number
c        iprop    (int,sc,in)       - property reference number:
c        ---- MP command labels --------
c        EX  = 1, EY  = 2, EZ  = 3, NUXY= 4, NUYZ= 5, NUXZ= 6, GXY = 7, GYZ = 8
c        GXZ = 9, ALPX=10, ALPY=11, ALPZ=12, DENS=13, MU  =14, DAMP=15, KXX =16
c        KYY =17, KZZ =18, RSVX=19, RSVY=20, RSVZ=21, C   =22, HF  =23, VISC=24
c        EMIS=25, ENTH=26, LSST=27, PRXY=28, PRYZ=29, PRXZ=30, MURX=31, MURY=32
c        MURZ=33, PERX=34, PERY=35, PERZ=36, MGXX=37, MGYY=38, MGZZ=39, EGXX=40
c        EGYY=41, EGZZ=42, SBKX=43, SBKY=44, SBKZ=45, SONC=46, DMPS=47, ELIM=48
c        USR1=49, USR2=50, USR3=51, USR4=51, FLUI=53, ORTH=54, CABL=55, RIGI=56
c        HGLS=57, BVIS=58, QRAT=59, REFT=60, CTEX=61, CTEY=62, CTEZ=63, THSX=64,
c        THSY=65, THSZ=66, DMPR=67, LSSM=68, BETD=69, ALPD=70, RH =71,  DXX =72,
c        DYY =73, DZZ =74, BETX=75, BETY=76, BETZ=77, CSAT=78, CREF=79, CVH =80,
c        UMID=81, UVID=82
c
c                   (see mpinit for uncommented code and TB command information)

c     output arguments:
c        mpget   (int,func,out)     - number of temperature values
c        temp    (dp,ar(mpget),out) - vector of the temperature values
c        prop    (dp,ar(mpget),out) - vector of the property values
```

3.3.11. Subroutine mpput (Storing a Material Property Table)

```
*deck,mpput
      subroutine mpput (mat,iprop,ntab,temp,prop)
c *** primary function:    store material property tables.

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        mat      (int,sc,in)       - material number.
c        iprop    (int,sc,in)       - property reference number:
c        ---- MP command labels --------
c        EX  = 1, EY  = 2, EZ  = 3, NUXY= 4, NUYZ= 5, NUXZ= 6, GXY = 7, GYZ = 8
c        GXZ = 9, ALPX=10, ALPY=11, ALPZ=12, DENS=13, MU  =14, DAMP=15, KXX =16
```

```
c        KYY =17, KZZ =18, RSVX=19, RSVY=20, RSVZ=21, C   =22, HF  =23, VISC=24
c        EMIS=25, ENTH=26, LSST=27, PRXY=28, PRYZ=29, PRXZ=30, MURX=31, MURY=32
c        MURZ=33, PERX=34, PERY=35, PERZ=36, MGXX=37, MGYY=38, MGZZ=39, EGXX=40
c        EGYY=41, EGZZ=42, SBKX=43, SBKY=44, SBKZ=45, SONC=46, DMPS=47, ELIM=48
c        USR1=49, USR2=50, USR3=51, USR4=51, FLUI=53, ORTH=54, CABL=55, RIGI=56
c        HGLS=57, BVIS=58, QRAT=59, REFT=60, CTEX=61, CTEY=62, CTEZ=63, THSX=64,
c        THSY=65, THSZ=66, DMPR=67, LSSM=68, BETD=69, ALPD=70, RH  =71, DXX =72,
c        DYY =73, DZZ =74, BETX=75, BETY=76, BETZ=77, CSAT=78, CREF=79, CVH =80,
c        UMID=81, UVID=82
c
c                  (see mpinit for uncommented code and TB command information)

c        ntab     (int,sc,in)       - number of entries in the table
c                                      (1 to 100)
c        tem     (dp,ar(ntab),in)  - temperature vector (ascending)
c        prp     (dp,ar(ntab),in)  - property vector

c     output arguments:
c        none.
```

3.3.12. Subroutine mpdel (Deleting a Material Property Table)

```
*deck,mpdel
      subroutine mpdel (mat,iprop)
c *** primary function:    delete material property tables.

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        mat      (int,sc,in)       - material number.
c        iprop    (int,sc,in)       - property reference number:
c                                      (0 = all properties)
c        ---- MP command labels --------
c        EX  = 1, EY  = 2, EZ  = 3, NUXY= 4, NUYZ= 5, NUXZ= 6, GXY = 7, GYZ = 8
c        GXZ = 9, ALPX=10, ALPY=11, ALPZ=12, DENS=13, MU  =14, DAMP=15, KXX =16
c        KYY =17, KZZ =18, RSVX=19, RSVY=20, RSVZ=21, C   =22, HF  =23, VISC=24
c        EMIS=25, ENTH=26, LSST=27, PRXY=28, PRYZ=29, PRXZ=30, MURX=31, MURY=32
c        MURZ=33, PERX=34, PERY=35, PERZ=36, MGXX=37, MGYY=38, MGZZ=39, EGXX=40
c        EGYY=41, EGZZ=42, SBKX=43, SBKY=44, SBKZ=45, SONC=46, DMPS=47, ELIM=48
c        USR1=49, USR2=50, USR3=51, USR4=51, FLUI=53, ORTH=54, CABL=55, RIGI=56
c        HGLS=57, BVIS=58, QRAT=59, REFT=60, CTEX=61, CTEY=62, CTEZ=63, THSX=64,
c        THSY=65, THSZ=66, DMPR=67, LSSM=68, BETD=69, ALPD=70, RH  =71, DXX =72,
c        DYY =73, DZZ =74, BETX=75, BETY=76, BETZ=77, CSAT=78, CREF=79, CVH =80,
c        UMID=81, UVID=82
c
c               (see mpinit for uncommented code and for TB command information)

c     output arguments:  none.
```

3.3.13. Function rlinqr (Getting Information About a Real Constant Set)

```
*deck,rlinqr
      function rlinqr (nreal,key)
c *** primary function:    get information about a real constant set
c *** secondary functions: none

c *** Notice - This file contains ANSYS Confidential information ***

c     typ=int,dp,log,chr,dcp   siz=sc,ar(n),func    intent=in,out,inout

c  input arguments:
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c        variable (typ,siz,intent)    description
c        nreal   (int,sc,in)       - real constant table number
c                                      should be 0 for key=11, DB_NUMDEFINED,
c                                      DB_NUMSELECTED, DB_MAXDEFINED, and
c                                      DB_MAXRECLENG
c        key      (int,sc,in)       - information flag.
c             = 5              - return number of values stored for nreal
c             = DB_SELECTED    - return select status
c                      rlinqr = 0 - real constant table is undefined.
c                             =-1 - real constant table is unselected.
c                             = 1 - real constant table is selected
c             = DB_NUMDEFINED  - return number of defined real constant tables
c             = DB_NUMSELECTED - return number of selected real constant tables
c             = DB_MAXDEFINED  - return highest real constant table defined
c             = DB_MAXRECLENG  - return maximum record length (dp words)

c  output arguments:
c        rlinqr   (int,func,out)    - the returned value of rlinqr is based on
c                                     setting of key.
c *** mpg magnetic interface usage
c
```

3.3.14. Function rlget (Getting Real Constant Data)

```
*deck,rlget
      function rlget (nreal,rtable)
c *** primary function:    get real constant data

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        nreal    (int,sc,in)       - real constant table number

c     output arguments:
c        rlget    (int,func,out)    - number of real constant data obtained
c        rtable   (dp,ar(*),out)    - real constant data obtained
```

3.3.15. Subroutine rlsel (Selecting or Deleting a Real Constant Set)

```
*deck,rlsel
      subroutine rlsel (nreai,ksel)
c *** primary function:    select or delete a real constant set
c *** secondary functions: none
c
c *** Notice - This file contains ANSYS Confidential information ***
c
c     typ=int,dp,log,chr,dcp   siz=sc,ar(n),func    intent=in,out,inout
c
c  input arguments:
c     variable (typ,siz,intent)    description
c     nreai    (int,sc,in)       - real constant table
c                                    = 0 - all real constant tables
c     ksel     (int,sc,in)       - type of operation to be performed.
c                                    = 0 - delete real constant table.
c                                    = 1 - select real constant table.
c                                    =-1 - unselect real constant table.
c                                    = 2 - invert real constant table.
c
c  output arguments:
c     none
c
```

3.3.16. Function csyiqr (Getting Information About a Coordinate System)

```
*deck,csyiqr
      function csyiqr (ncsy,key)
c *** primary function:    get information about a coordinate system

c *** Notice - This file contains ANSYS Confidential information ***

c  input arguments:
c     ncsy     (int,sc,in)       - coordinate system reference number
c                                    should be zero for key= DB_NUMDEFINED
c                                    or DB_MAXDEFINED
c     key      (int,sc,in)       - information flag.
c              = DB_SELECTED    - return status:
c                           csyiqr = 0 - coordinate system is not defined
c                                   -1 - coordinate system is not selected
c                                    1 - coordinate system is selected
c              = DB_NUMDEFINED  - number of defined coordinate systems
c              = DB_MAXDEFINED  - maximum coordinate system reference
c                                 number used.

c  output arguments:
c     csyiqr   (int,func,out)    - the returned value of csyiqr is based on
c                                    setting of key.
```

3.3.17. Function csyget (Getting a Coordinate System)

```
*deck,csyget
      function csyget (ncsy,csydpx,csyinx)
c *** primary function:    get a coordinate system
c *** secondary functions: none

c *** Notice - This file contains ANSYS Confidential information ***

c  NOTE:  As a time-saving device, this routine will not fetch the coordinate
c         system data from the database (an expensive operation)
c         if ncsy = csyinx(4), as this would indicate that the data is current.
c         If you wish to force this routine to fetch coordinate system data (in
c         the case of loading a local array, for example), you MUST set
c         ncsy != csyinx(4) before function call.

c     typ=int,dp,log,chr,dcp   siz=sc,ar(n),func    intent=in,out,inout

c  input arguments:
c     variable (typ,siz,intent)    description                 csycom name
c     ncsy     (int,sc,in)       - coordinate system number
c     csyinx(4) (int,sc,inout)   - coordinate system number     csyact

c  output arguments:
c     csydpx   (dp,ar(18),out)
c                           csydpx(1-9)   - transformation matrix
c                                 (10-12) - origin (XC, YC, ZC)
c                                 (13-14) - coordinate system parameters  cparm
c                                                                         cparm2
c                                 (15)    - spare
c                                 (16-18) - defining angles
c     csyinx   (int,ar(6),out)
c                           csyinx(1-2)   - theta, phi singularity keys
c                                 (3)     - coordinate system type        icdsys
c         (csyinx(4) is inout)  (4)     - coordinate system number      csyact
c                                 (5)     - spare
c                                 (6)     - spare
c     csyget   (int,func,out)      - status of coordinate system
c                                    = 0 - coordinate system exists
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                  = 1 - coordinate system doesn't exist
```

3.3.18. Subroutine csyput (Storing a Coordinate System)

```
*deck,csyput
      subroutine csyput (ncsy,csydpx,csyinx)
c *** primary function:    store a coordinate system

c *** Notice - This file contains ANSYS Confidential information ***

c  input arguments:
c     ncsy     (int,sc,in)       - coordinate system number
c     csydpx   (dp,ar(18),out)
c                         csydpx(1-9)   - transformation matrix
c                               (10-12) - origin (XC, YC, ZC)
c                               (13-14) - coordinate system parameters     cparm
c                                                                        cparm2
c                               (15)    - spare
c                               (16-18) - defining angles
c     csyinx   (int,ar(6),out)
c                         csyinx(1-2) - theta, phi singularity keys
c                               (3)   - coordinate system type           icdsys
c                               (4)   - coordinate system number         csyact
c                               (5)   - spare
c                               (6)   - spare


c  output arguments: none
```

3.3.19. Subroutine csydel (Deleting a Coordinate System)

```
*deck,csydel
      subroutine csydel (ncsy)
c *** primary function:    delete a coordinate system
c *** secondary functions: none
c
c     typ=int,dp,log,chr,dcp   siz=sc,ar(n),func    intent=in,out,inout
c
c *** Notice - This file contains ANSYS Confidential information ***
c
c  input arguments:
c     variable (typ,siz,intent)    description
c     ncsy     (int,sc,in)       - coordinate system number
c
c  output arguments:
c     none
c
```

3.3.20. Subroutine userac (Demonstrates Use of Element Attribute Routines)

See Subroutine userac (Accessing Element Information) (p. 175) for an example that demonstrates how to use the `userac` subroutine to extract information about an element type and element real constants from the Mechanical APDL database. You can find this subroutine on your product-distribution media.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.4. Coupling and Constraint Routines

3.4.1. Function cpinqr (Getting Information About a Coupled Set)

```
*deck,cpinqr
      function cpinqr (ncp,key)
c *** primary function:    get information about a coupled set
c *** secondary functions: none
c
c *** Notice - This file contains ANSYS Confidential information ***
c
c     typ=int,dp,log,chr,dcp   siz=sc,ar(n),func    intent=in,out,inout
c
c  input arguments:
c     variable  (typ,siz,intent)    description
c     ncp       (int,sc,in)     - coupled set number
c     key       (int,sc,in)     - inquiry key:
c                                   should be zero for key=11, DB_NUMDEFINED,
c                                    DB_NUMSELECTED, DB_MAXDEFINED, and
c                                    DB_MAXRECLENG
c                = DB_SELECTED    - return select status
c                            cpinqr = 1 - coupled set is selected
c                                   = 0 - coupled set in undefined
c                                   =-1 - coupled set in unselected
c                = DB_NUMDEFINED  - return number of defined coupled sets
c                = DB_NUMSELECTED - return number of selected coupled sets
c                = DB_MAXDEFINED  - return the number of the highest numbered
c                                    coupled set
c                = DB_MAXRECLENG  - return length of largest coupled set record
c                                    (max record length)
c                =  2             - return length (data units)
c                =  3             - return layer number
c                =  4             - return address of first data word
c                =  5             - return number of values stored for ncp
c                = 11             - return void percent (integer)
c                = 16             - return location of next record
c                = -1             - return master node for this eqn (this is
c                                   currently only used by solution DB object)
c
c  output arguments:
c     cpinqr   (int,func,out)    - the returned value of cpinqr is based on
c                                        setting of key
c
```

3.4.2. Function cpget (Getting a Coupled Set)

```
*deck,cpget
      function cpget (ncp,ieqn)
c *** primary function:    get a coupled set

c *** Notice - This file contains ANSYS Confidential information ***

c  input arguments:
c     ncp      (int,sc,in)                 - coupled set number
c
c  output arguments:
c     cpget         (int,func,out)        - number of nodes in list
c     ieqn          (int,ar(cpget+2),out) - coupled set info:
c                              ieqn(1:cpget) - list of coupled nodes
c                              ieqn(cpget+1) - set degree of freedom
c                              ieqn(cpget+2) - number of nodes in list
c                                              (copy of return value)
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.4.3. Subroutine cpput (Storing a Coupled Set)

```
*deck,cpput
      subroutine cpput (ncp,n,ieqn)
c *** primary function:    store a coupling set

c *** Notice - This file contains ANSYS Confidential information ***

c  input arguments:
c     ncp       (int,sc,in)        - coupled set number
c     n         (int,sc,in)        - number of nodes in coupled set
c     ieqn      (int,ar(n+2),in)   - info for storage
c                            ieqn(1:n) - list of coupled nodes
c                            ieqn(n+1) - degree of freedom label for set
c    (ieqn(n+2) is inout)    ieqn(n+2) - number of nodes in coupled set
c                                        (copy of n)

c  output arguments:
c     ieqn(n+2) (int,sc,inout)     - number of nodes in coupled set
c                                    (another copy of n)
```

3.4.4. Subroutine cpsel (Selecting or Deleting a Coupled Set)

```
*deck,cpsel
      subroutine cpsel (ncpi,ksel)
c *** primary function:    select or delete a coupled set
c *** secondary functions: none
c
c *** Notice - This file contains ANSYS Confidential information ***
c
c     typ=int,dp,log,chr,dcp   siz=sc,ar(n),func    intent=in,out,inout
c
c  input arguments:
c     variable  (typ,siz,intent)    description
c     ncpi      (int,sc,in)       - coupled set number
c     ksel      (int,sc,in)       - select/delete flag
c                                   = 0 - delete coupled set
c                                   = 1 - select coupled set
c  output arguments:
c     none
c
```

3.4.5. Function ceinqr (Getting Information About a Constraint Equation Set)

```
*deck,ceinqr
      function ceinqr (nce,key)
c *** primary function:    get information about a constraint equation set

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c     nce         (int,sc,in)       - constraint equation number
c     key         (int,sc,in)       - inquiry key:
c                                     should be zero for key=11, DB_NUMDEFINED,
```

```
c                                       DB_NUMSELECTED, DB_MAXDEFINED, and
c                                       DB_MAXRECLENG
c            = DB_SELECTED    - return select status
c                                    ceinqr = 1 - equation is selected
c                                           = 0 - equation is undefined
c                                           =-1 - equation is unselected
c            = DB_NUMDEFINED  - return number of defined contraint equations
c            = DB_NUMSELECTED - return number of selected contraint equations
c            = DB_MAXDEFINED  - return number of highest numbered constraint
c                               equation defined
c            = DB_MAXRECLENG  - return length of longest contraint equation set
c                               (max record length)
c            =  2             - return length (data units)
c            =  3             - return layer number
c            =  4             - address of first data word
c            =  5             - return number of values stored for nce
c            = 11             - return void percent (integer)
c            = 16             - return location of next record
c            = CE_NONLINEAR   - return 1 if CE is nonlinear
c            = CE_ELEMNUMBER  - return associated element number

c     output arguments:
c        ceinqr   (int,func,out)    - the returned value of ceinqr is based on
c                                     setting of key
```

3.4.6. Function ceget (Getting a Constraint Equation)

```
*deck,ceget
      function ceget (nce,ieqn,deqn)
c *** primary function:    get a constraint equation

c *** Notice - This file contains ANSYS Confidential information ***
c     input arguments:
c        nce     (int,sc,in)      - constraint equation set number
c     output arguments:
c        ieqn    (int,ar(2,*),out)      ieqn(1:2,1:n) - node, dof
c                                        ieqn(2,n+1) - number of dof in set
c                                        NOTE: negative means internal CE
c        deqn    (dp,ar(*),out)         deqn(1:n) - coefficients of
c                                                   each dof in set
c                                        deqn(n+1) - constant term
```

3.4.7. Subroutine ceput (Storing a Constraint Equation)

```
*deck,ceput
      subroutine ceput (nce,n,ieqn,deqn)
c *** primary function:    store a constraint equation

c *** Notice - This file contains ANSYS Confidential information ***
c     input arguments:
c        nce     (int,sc,in)            - constraint equation set number
c     output arguments:
c        n       (int,sc,inout)         - number of degrees of freedom in set
c        ieqn    (int,ar(2,n+1),inout)  - ieqn(1:2,1:n) - node, dof
c                                        - ieqn(2,n+1) - number of dof in set
c                                        Note: negative means internal CE
c
c        deqn    (dp,ar(n+1),inout)    - deqn(1:n) coefficients of each dof in set
c                                        - deqn(n+1) - constant term
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.4.8. Subroutine cesel (Deleting or Selecting a Constraint Equation)

```
*deck,cesel
      subroutine cesel (ncei,ksel)
c *** primary function:    select or delete a constraint equation

c *** Notice - This file contains ANSYS Confidential information ***

c *** mpg cesel cedele < pr7rst, edgung solvcl  - delete ce
c     input arguments:
c      ncei  (int,sc,in)             - constraint equation number
c      ksel  (int,sc,in)             - select/delete flag
c                                      = 0 - delete equation
c     output arguments:
c         none
```

3.5. Nodal Loading Routines

The following table provides available body load keys for the routines used to access nodal body load information (`ansNodeBodyLoadIqr`, `ansNodeBodyLoadGet`, `ansNodeBodyLoadPut`, and `ansNodeBodyLoadDel`).

Table 3.1: Nodal Body Load Keys

Key	Internal Label	BF Cmd Label	Meaning
1	BODYLOAD_TEMP	TEMP	Temperature
2	BODYLOAD_FLUE	FLUE	Fluence
3	BODYLOAD_HGEN	HGEN	Heat generation
4	BODYLOAD_DGEN	DGEN	Diffusing substance genereation rate
5	BODYLOAD_MVDI	MVDI	Magnetic virtual displacement flags
6	BODYLOAD_CHRG	CHRDG	Magnetic virtual displacement flags
11	BODYLOAD_PORT	PORT	Interior waveguide/transmission line port
14	BODYLOAD_VMEN	VMEN	Mean flow
16	BODYLOAD_IMPD	IMPD	Impedance sheet
18	BODYLOAD_FSOU	FSOU	Fluid flow source
20	BODYLOAD_VELO	VELO	Velocity
21	BODYLOAD_MASS	MASS	Mass source
22	BODYLOAD_SPRE	SPRE	Static pressure
23	BODYLOAD_FPBC	FPBC	Flouqet periodic phase
25	BODYLOAD_UFOR	UFOR	Force potential (same slot as EF)
26	BODYLOAD_HFLW	HFLW	Heat flow (same slot as EF)
27	BODYLOAD_SFOR	SFOR	Shear force (same slot as VELO)

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.5.1. Function disiqr (Getting Information About Constraints)

```
*deck,disiqr
      function disiqr (node,key)
c *** primary function: get information about constraints

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        node    (int,sc,in)       - node number for inquire.
c        key     (int,sc,in)       - key as to the information needed
c                          = 1             - return constraint mask
c                          = DB_MAXDEFINED,
c                            DB_NUMDEFINED  - return number of nodal constraints
c                                             NOTE: both DB_MAXDEFINED and
c                                             DB_NUMDEFINED produce the same
c                                             functionality

c     output arguments:
c        disiqr   (int,func,out)   - the returned value of disiqr is based on
c                                    setting of key.
```

3.5.2. Function disget (Getting a Constraint at a Node)

```
*deck,disget
      function disget (inode,idf,value)
c *** primary function:    get a constraint from the data base (in raw form)

c *** Notice - This file contains ANSYS Confidential information ***
c     input arguments:
c     variable (typ,siz,intent)     description
c        inode    (int,sc,in)       - node number (negative value for no
c                                                   partabeval)
c        idf      (int,sc,in)       - reference number for the DOF: (1-32)
c    UX  = 1, UY  = 2, UZ  = 3, ROTX= 4, ROTY= 5, ROTZ= 6, AX  = 7, AY  = 8
c    AZ  = 9, VX  =10, VY  =11, VZ  =12, GFV1=13, GFV2=14, GFV3=15, WARP=16
c    CONC=17, HDSP=18, PRES=19, TEMP=20, VOLT=21, MAG =22, ENKE=23, ENDS=24
c    EMF =25, CURR=26, SP01=27, SP02=28, SP03=29, SP04=30, SP05=31, SP06=32
c                                  (missing entries are spares)

c     output arguments:
c        disget   (int,func,out)    - status of constraint.
c                                     = 0 - no constraint on this node
c                                           for this DOF
c                                     = 4 - this node has a constraint
c                                           defined for this DOF
c                                     = -4 - this node has a pseudo-support
c                                           defined for this DOF
c        value    (dp,ar(4),out)    - constraint values
c                          value(1-2) - (real,imag) values of present settings
c                          value(3-4) - (real,imag) values of previous settings
```

3.5.3. Subroutine disput (Storing a Constraint at a Node)

```
*deck,disput
      subroutine disput (node,idf,value)
c *** primary function:    store a constraint at a node.

c *** Notice - This file contains ANSYS Confidential information ***
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c     input arguments:
c        node     (int,sc,in)       - node number
c        idf      (int,sc,in)       - reference number of DOF: (1-32)
c         UX  = 1, UY  = 2, UZ  = 3, ROTX= 4, ROTY= 5, ROTZ= 6, AX  = 7, AY  = 8
c         AZ  = 9, VX  =10, VY  =11, VZ  =12, GFV1=13, GFV2=14, GFV3=15, WARP=16
c         CONC=17, HDSP=18, PRES=19, TEMP=20, VOLT=21, MAG =22, ENKE=23, ENDS=24
c         EMF =25, CURR=26, SP01=27, SP02=28, SP03=29, SP04=30, SP05=31, SP06=32

c        value    (dp,ar(2),in)     - (real,imag) values for constraint

c     output arguments:  none.
```

3.5.4. Subroutine disdel (Deleting a Constraint at a Node)

```
*deck,disdel
      subroutine disdel (node,idf)
c *** primary function:    delete a constraint at a node

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        node     (int,sc,in)       - node number.
c        idf      (int,sc,in)       - reference number of DOF: (1-32)
c         UX  = 1, UY  = 2, UZ  = 3, ROTX= 4, ROTY= 5, ROTZ= 6, AX  = 7, AY  = 8
c         AZ  = 9, VX  =10, VY  =11, VZ  =12, GFV1=13, GFV2=14, GFV3=15, WARP=16
c         CONC=17, HDSP=18, PRES=19, TEMP=20, VOLT=21, MAG =22, ENKE=23, ENDS=24
c         EMF =25, CURR=26, SP01=27, SP02=28, SP03=29, SP04=30, SP05=31, SP06=32

c     output arguments:  none.
```

3.5.5. Function foriqr (Getting Information About Nodal Loads)

```
*deck,foriqr
      function foriqr (node,key)
c *** primary function: get information about nodal loads.

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        node     (int,sc,in)       - number of node being inquired about.
c                                     should be 0 for key=DB_MAXDEFINED or
c                                     DB_NUMDEFINED
c        key      (dp,sc,in)        - key as to information needed
c                                = 1             - return force mask for node
c                                = DB_MAXDEFINED,
c                                  DB_NUMDEFINED  - return number of nodal loadings
c                                                   in model
c                                     NOTE: both DB_MAXDEFINED and DB_NUMDEFINED
c                                     produce the same functionality

c     output arguments:
c        foriqr   (int,func,out)    - the returned value of foriqr is based on
c                                     setting of key.
```

3.5.6. Function forget (Getting a Nodal Load at a Node)

```
*deck,forget
      function forget (inode,idf,value)
c *** primary function:    get a force from the data base (in raw form)

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        inode    (int,sc,in)       - node number (negative value for no
c                                                  partabeval)
c        idf      (int,sc,in)       - reference number for the DOF: (1-32)
c                                     (see echprm.inc)

c     output arguments:
c        forget   (int,func,out)  - status of constraint.
c                                   = 0 - no loading on this node for this DOF
c                                   = 4 - this node has a loading for this DOF
c        value    (dp,ar(4),out)
c                          value(1-2) - (real,imag) values of present settings
c                          value(3-4) - (real,imag) values of previous settings
```

3.5.7. Subroutine forput (Storing a Nodal Load at a Node)

```
*deck,forput
      subroutine forput (node,idf,value)
c *** primary function:    store a nodal load at a node

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        node     (int,sc,in)       - node number
c        idf      (int,sc,in)       - reference number for the DOF: (1-32)
c         FX  = 1, FY  = 2, FZ  = 3, MX  = 4, MY  = 5, MZ  = 6, CSGX= 7, CSGY= 8
c         CSGZ= 9, VFX =10, VFY =11, VFZ =12
c         RATE=17           FLOW=19, HEAT=20, AMPS=21, FLUX=22, NPKE=23, NPDS=24
c         CURT=25, VLTG=26           (missing entries are spares)

c        value    (dp,ar(2),in)     - (real,imag) values of force

c     output arguments:  none.
```

3.5.8. Subroutine fordel (Deleting a Nodal Load at a Node)

```
*deck,fordel
      subroutine fordel (node,idf)
c *** primary function:    delete a nodal load at a node
c *** secondary functions: none.

c *** Notice - This file contains ANSYS Confidential information ***

c     typ=int,dp,log,chr,dcp   siz=sc,ar(n),func    intent=in,out,inout

c     input arguments:
c     variable (typ,siz,intent)     description
c        node     (int,sc,in)       - node number
c        idf      (int,sc,in)       - reference number for the DOF: (1-32)
c         FX  = 1, FY  = 2, FZ  = 3, MX  = 4, MY  = 5, MZ  = 6, CSGX= 7, CSGY= 8
c         CSGZ= 9, VFX =10, VFY =11, VFZ =12
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c         RATE=17,          FLOW=19, HEAT=20, AMPS=21, FLUX=22, NPKE=23, NPDS=24
c         CURT=25, VLTG=26           (missing entries are spares)

c     output arguments:
c        none.
```

3.5.9. Function ansNodeBodyLoadIqr (Getting Information About a Nodal Body Load)

```
*deck,ansNodeBodyLoadIqr
      function ansNodeBodyLoadIqr (BODYLOAD_KEY,node,key)
c *** primary function:    get information about a nodal body load

c *** Notice - This file contains ANSYS Confidential information ***

c     typ=int,dp,log,chr,dcp   siz=sc,ar(n),func    intent=in,out,inout

c input arguments:
c  variable (typ,siz,intent)    description
c     BODYLOAD_KEY  (int,sc,in)     - body load key (ansysdef.inc)
c     node          (int,sc,in)     - node number
c                                     should be zero for key=2
c     key           (int,sc,in)     - key for operation
c                                     = 1 - return nodal body load status
c                                     ansNodeBodyLoadIqr
c                                     = 0 - node has no body load defined
c                                     = 1 - node has a body load defined
c                                     = 2 - return total number of nodal body
c                                           load with BODYLOAD_KEY defined in model

c output arguments:
c  ansNodeBodyLoadIqr (int,func,out) - the returned value of ansNodeBodyLoadIqr
c                                   z   is based on setting of key.
```

3.5.10. Function ansNodeBodyLoadGet (Getting a Nodal Body Load Value)

```
*deck,ansNodeBodyLoadGet
      function ansNodeBodyLoadGet (BODYLOAD_KEY,node,val)
c *** primary function:    get specified nodal load

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        BODYLOAD_KEY       (int,sc,in)     - body load key (see ansysdef.inc)
c        node               (int,sc,in)     - node number

c     output arguments:
c        ansNodeBodyLoadGet (int,func,out)  - nodal load status of node.
c                                             = 0 - nodal load undefined
c                                             = 1 - nodal load is defined
c        val                (dp,ar(*),out)  - the nodal load (new,old)
```

3.5.11. Subroutine ansNodeBodyLoadPut (Storing a Nodal Body Load)

```
*deck,ansNodeBodyLoadPut
      subroutine ansNodeBodyLoadPut (BODYLOAD_KEY,node,nval,val)
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c *** primary function:    store nodal body loads

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        BODYLOAD_KEY(int,sc,in)       - body load key (see ansysdef.inc)
c        node        (int,sc,in)       - node number
c        nval        (int,sc,in)       - number of values to put
c        val         (dp ,ar(*),in)    - nodal loads

c     output arguments:  none.
```

3.5.12. Subroutine ansNodeBodyLoadDel (Deleting a Nodal Body Load)

```
*deck,ansNodeBodyLoadDel
      subroutine ansNodeBodyLoadDel (BODYLOAD_KEY,node)
c *** primary function:    delete node body loads

c *** Notice - This file contains ANSYS Confidential information ***
c
c     input arguments:
c     variable (typ,siz,intent)    description
c        BODYLOAD_KEY   (int,sc,in)       - body load key (see ansysdef.inc)
c        node           (int,sc,in)       - node number
c
c     output arguments:
c        none
c
```

3.6. Element Loading Routines

The following table provides available body load keys for the routines used to access element body load information (`ansElemBodyLoadIqr`, `ansElemBodyLoadGet`, `ansElemBodyLoadPut`, and `ansElemBodyLoadDel`).

Table 3.2: Element Body Load Keys

Key	Internal Label	BFE Cmd Label	Meaning
1	BODYLOAD_TEMP	TEMP	Temperature
2	BODYLOAD_FLUE	FLUE	Fluence
3	BODYLOAD_HGEN	HGEN	Heat generation
4	BODYLOAD_DGEN	DGEN	Diffusing substance genereation rate
5	BODYLOAD_MVDI	MVDI	Magnetic virtual displacement flags
6	BODYLOAD_CHRG	CHRDG	Magnetic virtual displacement flags
7	BODYLOAD_JS	JS	Mass source/Current density
9	BODYLOAD_EF	EF	Electric field
12	BODYLOAD_FVIN	FVIN	Field volume interface flag
18	BODYLOAD_FSOU	FSOU	Fluid flow source

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Key	Internal Label	BFE Cmd Label	Meaning
19	BODYLOAD_FORC	FORC	Body force density in momentum equation

The following table provides available surface load keys for the routines used to access element surface load information (`ansElemSurfLoadIqr`, `ansElemSurfLoadGet`, `ansElemSurfLoadPut`, and `ansElemSurfLoadDel`).

Table 3.3: Element Surface Load Keys

Key	Internal Label	Cmd Label	SF Cmd	SFE Cmd	Meaning
LOADS					
1	SURFLOAD_PRES	PRES	x	x	Pressure
2	SURFLOAD_CONV	CONV	x	x	Convection
3	SURFLOAD_IMPD	IMPD	x	x	Impedence
4	SURFLOAD_DFLU	DFLU	x	x	Diffusing substance generation rate
6	SURFLOAD_CHRG	CHRG	x	x	Charge density
7	SURFLOAD_PORT	PORT	x	x	Port
8	SURFLOAD_RAD	RAD	x	x	Radiation boundary
9	SURFLOAD_FSIN	FSIN	x	x	FSI interface
10	SURFLOAD_RDSF	RDSF	x	x	Surface-to-surface radiation
11	SURFLOAD_SELV	SELV		x	Substructure
12	SURFLOAD_SHLD	SHLD	x	x	Surface normal velocity or acceleration
13	SURFLOAD_FFLX	FFLX	x	x	Flow flux
16	SURFLOAD_HFLU	HFLU	x	x	Heat flux (film coefficient in db)
17	SURFLOAD_ATTN	ATTN	x	x	Attenuation coefficient
18	SURFLOAD_TIMP	TIMP	x		Thermal impedance
19	SURFLOAD_VIMP	VIMP	x		Viscous impedance
FLAGS					
1	SURFLOAD_MXWF	MXWF	x	x	Maxwell force
2	SURFLOAD_FREE	FREE	x	x	Free surface
3	SURFLOAD_INF	INF	x	x	Infinite surface
4	SURFLOAD_BLI	BLI	x		Boundary layer
5	SURFLOAD_FSI	FSI	x	x	FSI interface
9	SURFLOAD_RIGW	RIGW	x		Rigid wall

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.6.1. Function ansElemBodyLoadIqr (Getting Information About an Element Body Load)

```
*deck,ansElemBodyLoadIqr
      function ansElemBodyLoadIqr (BODYLOAD_KEY,ielem,key)
c *** primary function: get information about element loads.

c *** Notice - This file contains ANSYS Confidential information ***

c input arguments:
c  BODYLOAD_KEY (int,sc,in)     - body load key (see ansysdef.inc)
c  ielem        (int,sc,in)     - element number
c                                Should be 0 for key=11, DB_NUMDEFINED,
c                                DB_MAXDEFINED, and DB_MAXRECLENG
c  key          (int,sc,in)     - information flag.
c               = DB_SELECTED    - return status:
c                                  ansElemBodyLoadIqr = 0 - element has no loads
c                                                     = 1 - element has load defined
c               = DB_NUMDEFINED   - return number of loads defined for this element
c                                  (rec length)
c               = DB_MAXDEFINED   - return number of loads defined in model
c               = DB_MAXRECLENG   - return maximum number of loads defined for
c                                  any element (max rec length)
c               = 2               - return length (dp words)
c               = 3               - return layer number (for cross reference
c                                  files returnnumber of entities)
c               = 4               - return address of first data word
c               = 5               - return length (dp words)
c               = 6               - return compressed record number.
c               =11               - return void percent (integer)
c               =16               - return location of next record (this
c                                  increments the next record count)
c               =18               - return type of file.
c                                   ansElemBodyLoadIqr = 0 - integer
c                                                      = 1 - double precision
c                                                      = 2 - real
c                                                      = 3 - complex
c                                                      = 4 - character*8
c                                                      = 7 - index
c              =19               - return virtual type of file.
c                                  ansElemBodyLoadIqr = 0 - fixed length (4.4 form)
c                                                     = 1 - indexed variable length
c                                                           (layer data)
c                                                     = 2 - xref data tables
c                                                     = 3 - bitmap data
c                                                           (for 32 data item packed
c                                                            records)
c                                                     = 4 - data tables (three dimensional
c                                                           arrays)


c     output arguments:
c        ansElemBodyLoadIqr (int,func,out)    - the returned value of ansElemBodyLoadIqr is
c                                               based on setting of key.
```

3.6.2. Function ansElemBodyLoadGet (Getting an Element Body Load Value)

```
*deck,ansElemBodyLoadGet
      function ansElemBodyLoadGet (BODYLOAD_KEY,ielem,val)
c *** primary function:    get element loads (in raw form)

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c        BODYLOAD_KEY(int,sc,in)       - body load key (see ansysdef.inc)
c        ielem       (int,sc,in)       - element number

c     output arguments:
c        ansElemBodyLoadGet (int,func,out)    - status of element.
c                                               = 0 - no element load
c                                               > 0 - number of element load
c                                                     retrieved
c        val         (dp,ar(n,2),out)  - the element load (new,old).
```

3.6.3. Subroutine ansElemBodyLoadPut (Storing an Element Body Load)

```
*deck,ansElemBodyLoadPut
      subroutine ansElemBodyLoadPut (BODYLOAD_KEY,ielem,nVals,values)
c *** primary function:    store element loads

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        BODYLOAD_KEY(int,sc,in)       - body load key (see ansysdef.inc)
c        ielem       (int,sc,in)       - element number
c        nVals       (int,sc,in)       - number of element load values
c        values      (dp,ar(nVals),in) - element load
c
c     output arguments:  none
```

3.6.4. Subroutine ansElemBodyLoadDel (Deleting an Element Body Load)

```
*deck,ansElemBodyLoadDel
      subroutine ansElemBodyLoadDel (BODYLOAD_KEY,ielem)
c *** primary function:    delete element loads

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        BODYLOAD_KEY(int,sc,in)       - body load key (ansysdef.inc)
c        ielem       (int,sc,in)       - element number

c     output arguments:  none.
```

3.6.5. Function ansElemSurfLoadIqr (Getting Information About an Element Surface Load)

```
*deck,ansElemSurfLoadIqr
      function ansElemSurfLoadIqr (SURFLOAD_KEY,ielem,iface,key)
c *** primary function: get information about element pressure/convection

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        SURFLOAD_KEY(int,sc,in)       - surface load key
c        ielem       (int,sc,in)       - element number
c                                       should be zero for key=DB_NUMDEFINED or
c                                       DB_MAXRECLENG
c        iface       (int,sc,in)       - face number for inquire (0-6)
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                       face number is needed for key=5. for
c                                       other values of key, iface has different
c                                       meaning (see below)
c        key          (int,sc,in)       - key as to the information needed
c                     = 1               - return pressure mask for element
c                     = 5               - return number of pressures for this
c                                         element face. if face = 0,
c                                         returns max. data size
c                     = DB_NUMDEFINED,
c                     = DB_MAXDEFINED   - return value is based on setting of iface
c                                          NOTE: both DB_NUMDEFINED and
c                                                DB_MAXDEFINED produce the same
c                                                functionality
c                                 iface = 0   - return number of surface loads defined
c                                       = 1-6 - return number of pressure loads
c                                                defined for this element.
c                                                NOTE:  only 1-6 is valid, but this
c                                                       routine simply checks that iface
c                                                       is in the range.  The actual value
c                                                       of iface does not matter in this case.
c                    = DB_MAXRECLENG  - return the maximum number of element
c                                       pressures on any element (max record
c                                       length)

c     output arguments:
c        ansElemSurfLoadIqr(int,func,out)  - the returned value of ansElemSurfLoadIqr
c                                            is based on setting of key.
c
```

3.6.6. Function ansElemSurfLoadGet (Getting an Element Surface Load Value)

```
*deck,ansElemSurfLoadGet
      function ansElemSurfLoadGet (SURFLOAD_KEY,elem,iface,value)
c *** primary function:    get an element face load

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        SURFLOAD_KEY        (int,sc,in)       - surface load key
c        elem                (int,sc,in)       - element number (negative value for
c                                                                  no partabeval)
c        iface               (int,sc,in)       - face number (1-68)

c     output arguments:
c        ansElemSurfLoadGet  (int,func,out)    - status of element
c                                                  =-1 - element has no surface load of given type
c                                                  = 0 - element face has no surface load of given type
c                                                  > 0 - number of surface load values defined
c        value               (dp ,ar(*),out)   - the element surface load values (real,imag)
c                                                at the given face for the given type
```

3.6.7. Subroutine ansElemSurfLoadPut (Storing an Element Surface Load)

```
*deck,ansElemSurfLoadPut
      subroutine ansElemSurfLoadPut (SURFLOAD_KEY,ielem,iface,nval,
     x                               value)
c *** primary function:    store an element face loads.

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        SURFLOAD_KEY   (int,sc,in)       - surface load key
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c        ielem          (int,sc,in)       - element number for operation
c        iface          (int,sc,in)       - face number (1-68)
c        nval           (int,sc,in)       - number of values to put
c        value          (dp,ar(nval),in)  - the element load (real,imag) at each
c                                            face
c
c     output arguments:  none
```

3.6.8. Subroutine ansElemSurfLoadDel (Deleting an Element Surface Load)

```
*deck,ansElemSurfLoadDel
      subroutine ansElemSurfLoadDel (SURFLOAD_KEY,ielem,iface)
c *** primary function:    delete a surface load on an element

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        SURFLOAD_KEY   (int,sc,in)     - surface load key
c        ielem          (int,sc,in)     - element number
c        iface          (int,sc,in)     - face number
c                                          = 0 - delete all pressures on this
c                                                element
c                                          = 1-6 - delete pressure on this face
c     output arguments:  none.
c
```

3.7. Results Information Routines

3.7.1. Function dspiqr (Getting Information About Nodal Results)

```
*deck,dspiqr
      function dspiqr (node,key)
c *** primary function: get information about nodal results

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        node    (int,sc,in)       - node number
c                                    = 0 - return information based on the setting of key
c                                    > 0 - return result mask for given node
c        key     (int,sc,in)       - key as to the information needed
c                                      when node > 0 and key = 1 --> return result mask for given node
c                                      when node = 0 and key = DB_MAXRECLENG --> return maximum record length (dp
c                                                                                NOTE: only supported with databa
c                                      when node = 0 and key = any other value --> return number of calculated di

c     output arguments:
c        dspiqr   (int,func,out)   - the returned value of dspiqr is based on setting of node/key
```

3.7.2. Function dspget (Getting a Nodal Result from the Database)

```
*deck,dspget
      function dspget (node,ndf,idf,value)
c *** primary function:    get a nodal result from the data base
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        node     (int,sc,in)       - node number
c        ndf      (int,sc,in)       - number of results requested
c        idf      (int,ary(ndf),in) - reference number for the DOF: (1-32)
c    UX  = 1, UY  = 2, UZ  = 3, ROTX= 4, ROTY= 5, ROTZ= 6, AX  = 7, AY  = 8
c    AZ  = 9, VX  =10, VY  =11, VZ  =12, GFV1=13, GFV2=14, GFV3=15, WARP=16
c    CONC=17, HDSP=18, PRES=19, TEMP=20, VOLT=21, MAG =22, ENKE=23, ENDS=24
c    EMF =25, CURR=26, SP01=27, SP02=28, SP03=29, SP04=30, SP05=31, SP06=32
c                                (missing entries are spares)

c     output arguments:
c        value    (dp,ar(ndf),out)  - result values
```

3.7.3. Subroutine dspput (Storing a Result at a Node)

```
*deck,dspput
      subroutine dspput (node,ndf,idf,value)
c *** primary function:    store a result at a node.

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        node     (int,sc,in)       - node number
c        ndf      (int,sc,in)       - number of results to be stored
c        idf      (int,ary(ndf),in) - reference number for the DOF: (1-32)
c        value    (dp,ar(ndf),in)   -  displacement values

c     output arguments:  none
```

3.7.4. Subroutine dspdel (Deleting a Result at a Node)

```
*deck,dspdel
      subroutine dspdel (node,ndf,idf)
c *** primary function:    delete a result at a node

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        node     (int,sc,in)       - node number. (0 to delete DOF at all
c                                      nodes)
c        ndf      (int,sc,in)       - number of DOFs to delete (0 to delete
c                                      all DOFs)
c        idf      (int,ar(*),in)    - reference number for the DOF: (1-32)
c         UX  = 1, UY  = 2, UZ  = 3, ROTX= 4, ROTY= 5, ROTZ= 6, AX  = 7, AY  = 8
c         AZ  = 9, VX  =10, VY  =11, VZ  =12, GFV1=13, GFV2=14, GFV3=15, WARP=16
c         CONC=17, HDSP=18, PRES=19, TEMP=20, VOLT=21, MAG =22, ENKE=23, ENDS=24
c         EMF =25, CURR=26, SP01=27, SP02=28, SP03=29, SP04=30, SP05=31, SP06=32

c     output arguments:  none
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.7.5. Function emsiqr (Getting Information About an Element's Miscellaneous Summable Data)

```
*deck,emsiqr
      function emsiqr (ielem,key)
c *** primary function:    get information about element misc summable data

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem     (int,sc,in)      - element number (or zero, see below)
c        key     (int,sc,in)        - key as to the information needed
c                     =  1 - return info about misc summed data records
c                                    ielem > 0 - return number of misc summed
c                                                data items for this element
c                                                (record length)
c                                         = 0 - return maximum number of misc
c                                                summed data items on any
c                                                element (max record length)
c                     = DB_NUMDEFINED - return total number of misc summed data
c                                        items defined in model

c     output arguments:
c        emsiqr   (int,func,out)    - the returned value of emsiqr is based on
c                                     setting of key
```

3.7.6. Function emsget (Getting an Element's Miscellaneous Summable Data)

```
*deck,emsget
      function emsget (ielem,value)
c *** primary function:    get element misc summable data.

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem    (int,sc,in)       - element number

c     output arguments:
c        emsget   (int,func,out)    - status of element.
c                                     = 0 - element is undefined
c                                     > 0 - number of data items returned
c        value    (dp,ar(*),out)    - element misc summed data.

c                                    NOTE: the contents of this record is element
c                                        dependent.  See SMISC on ETABLE command
```

3.7.7. Subroutine emsput (Storing an Element's Miscellaneous Summable Data)

```
*deck,emsput
      subroutine emsput (ielem,nval,value)
c *** primary function:    store misc. summable data for an element.

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem     (int,sc,in)       - element number
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c        nval      (int,sc,in)       - number of values to be stored
c        value    (dp,ar(nval),in)   - the misc summed data values

c     output arguements:  none
c                                  NOTE: the contents of this record is element
c                                      dependent.  See SMISC on ETABLE command
```

3.7.8. Subroutine emsdel (Deleting an Element's Miscellaneous Summable Data)

```
*deck,emsdel
      subroutine emsdel (ielem)
c *** primary function:    delete element misc summable data

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem    (int,sc,in)       - element number
c                                     = 0 - delete for all defined elements

c     output arguments:  none.
```

3.7.9. Function enfiqr (Getting Information About Element Nodal Forces)

```
*deck,enfiqr
      function enfiqr (ielem,key)
c *** primary function: get information about element nodal forces

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem     (int,sc,in)      - element number (or zero, see below)
c        key     (int,sc,in)        - key as to the information needed
c                     =  1 - return info about element nodal forces
c                                    ielem > 0 - return number of element nodal
c                                                forces for this element
c                                                (record length)
c                                         = 0 - return maximum number of element
c                                                nodal forces on any element
c                                                (max record length)
c                     = DB_NUMDEFINED - return total number of element nodal
c                                       forces defined in model

c     output arguments:
c        enfiqr   (int,func,out)    - the returned value of enfiqr is based on
c                                     setting of key
```

3.7.10. Function enfget (Getting an Element's Nodal Forces)

```
*deck,enfget
      function enfget (ielem,value)
c *** primary function:    get element nodal forces.

c *** Notice - This file contains ANSYS Confidential information ***
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c     input arguments:
c        ielem    (int,sc,in)       - element number

c     output arguments:
c        enfget   (int,func,out)    - status of element.
c                                     = 0 - element has no nodal forces
c                                     > 0 - number of nodal forces returned
c        value    (dp,ar(*),out)    - element nodal forces
```

3.7.11. Subroutine enfput (Storing an Element's Nodal Forces)

```
*deck,enfput
      subroutine enfput (ielem,nval,value)
c *** primary function:    store nodal force results at an element.

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem     (int,sc,in)       - element number
c        nval      (int,sc,in)       - the total number of values
c                                       NOTE: There may be a maximum of 3 sets of
c                                     nodal forces in the record: static
c                                     forces, inertia forces, and damping forces
c        value    (dp,ar(nval),in)   - nodal force results

c     output arguments:  none
```

3.7.12. Subroutine enfdel (Deleting an Element's Nodal Forces)

```
*deck,enfdel
      subroutine enfdel (ielem)
c *** primary function:    delete element nodal forces data

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem    (int,sc,in)       - element number
c                                       = 0 - delete for all defined elements

c     output arguments:  none.
```

3.7.13. Function ensiqr (Getting Information About an Element's Nodal Stresses)

```
*deck,ensiqr
      function ensiqr (ielem,key)
c *** primary function: get information about element nodal stresses

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem     (int,sc,in)      - element number (or zero, see below)
c        key     (int,sc,in)        - key as to the information needed
c                    =  1 - return info about element nodal stresses
c                                    ielem > 0 - return number of element nodal
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                                 stresses for this element
c                                                 (record length)
c                                       = 0 - return maximum number of element
c                                                 nodal stresses on any element
c                                                 (max record length)
c                       = DB_NUMDEFINED - return total number of element
c                                         nodal stresses defined in model

c     output arguments:
c        ensiqr   (int,func,out)    - the returned value of ensiqr is based on
c                                     setting of key
```

3.7.14. Function ensget (Getting an Element's Nodal Stresses)

```
*deck,ensget
      function ensget (ielem,value)
c *** primary function:    get element nodal stresses.

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem    (int,sc,in)       - element number

c     output arguments:
c        ensget   (int,func,out)    - status of element.
c                                     = 0 - element undefined
c                                     > 0 - number of nodal stresses
c                                           returned
c        value    (dp,ar(*),out)    - element nodal stresses

c                                 NOTE: Stresses at each corner node in the order
c                                       X, Y, Z, XY, YZ, XZ, S1, S2, S3, SI, SE
c                                     For solid elements, stresses at each
c                                       corner node
c                                     For shell elements, stresses at each
c                                       corner node (first top durface, then
c                                       bottom)
c                                     For layered elements (w/KEYOPT(8)=0),
c                                       stresses for "first" layer at each
c                                       corner node (first at the bottom
c                                       surface of the bottom layer, then the
c                                       top surface of the top layer).
c                                       Stresses for "second" layer at each
c                                       corner node (first the bottom surface,
c                                       then the top surface for the layer with
c                                       the largest failure criteria).
c                                       The second layer is not present if
c                                       failure criteria were not used or are
c                                       not appropriate
c                                     For layered elements (w/KEYOPT(8)=1),
c                                       stresses for each layer at each corner
c                                       node (first at the bottom surface, then
c                                       the top surface)
c                                     For beam elements, the contents of this
c                                       record is element depenent.  See LS
c                                       item of ETABLE command.
```

3.7.15. Subroutine ensput (Storing Nodal Stresses at an Element)

```
*deck,ensput
      subroutine ensput (ielem,nval,value)
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c *** primary function:    store nodal stresses at an element.

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem     (int,sc,in)       - element number
c        nval      (int,sc,in)       - the total number of values
c                                       (11*nnod*nface)
c        value    (dp,ar(nval),in)   - the stress values

c     output arguments:  none

c                                NOTE: Stresses at each corner node in the order
c                                         X, Y, Z, XY, YZ, XZ, S1, S2, S3, SI, SE
c                                      For solid elements, stresses at each
c                                         corner node
c                                      For shell elements, stresses at each
c                                         corner node (first top surface, then
c                                         bottom)
c                                      For layered elements (w/KEYOPT(8)=0),
c                                         stresses for "first" layer at each
c                                         corner node (first at the bottom
c                                         surface of the bottom layer, then the
c                                         top surface of the top layer).
c                                         Stresses for "second" layer at each
c                                         corner node (first the bottom surface,
c                                         then the top surface for the layer with
c                                         the largest failure criteria).
c                                         The second layer is not present if
c                                         failure criteria were not used or are
c                                         not appropriate
c                                      For layered elements (w/KEYOPT(8)=1),
c                                         stresses for each layer at each corner
c                                         node (first at the bottom surface, then
c                                         the top surface)
c                                      For beam elements, the contents of this
c                                         record is element depenent.  See LS
c                                         item of ETABLE command.
```

3.7.16. Subroutine ensdel (Deleting an Element's Nodal Stresses)

```
*deck,ensdel
      subroutine ensdel (ielem)
c *** primary function:    delete element nodal stresses

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem    (int,sc,in)       - element number
c                                       = 0 - delete for all defined elements

c     output arguments:  none.
```

3.7.17. Function esfiqr (Getting Information About Element Surface Stress Data)

```
*deck,esfiqr
      function esfiqr (ielem,key)
c *** primary function:    get information about element surface stress data
```

```
c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem     (int,sc,in)      - element number (or zero, see below)
c        key     (int,sc,in)        - key as to the information needed
c                 = 1 - return info about surface stress
c                         ielem > 0 - return number of surface stresses on this
c                                     element (rec length)
c                               = 0 - return maximum number of surface stresses
c                                     on any element (max rec length)
c                 = DB_NUMDEFINED - return the number of surface stresses
c                                   defined in model

c     output arguments:
c        esfiqr   (int,func,out)    - the returned value of esfiqr is based on
c                                     setting of key
```

3.7.18. Function esfget (Getting Element Surface Stress Data)

```
*deck,esfget
      function esfget (ielem,value)
c *** primary function:    get element surface stress data.

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem    (int,sc,in)       - element number

c     output arguments:
c        esfget   (int,func,out)    - status of element.
c                                     = 0 - element undefined
c                                     > 0 - number of values returned
c        value    (dp,ar(*),out)    - element surface stress data.
```

3.7.19. Subroutine esfput (Storing Element Surface Stress Data)

```
*deck,esfput
      subroutine esfput (ielem,nval,value)
c *** primary function:    store surface stresses for an element.

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem     (int,sc,in)       - element number
c        nval      (int,sc,in)       - the total number of values
c                                      (19 * number of stress faces)
c                                      There is a max of 2 stress faces
c        value    (dp,ar(nval),in)   - the values

c     output arguments:  none
```

3.7.20. Subroutine esfdel (Deleting an Element's Surface Stress Data)

```
*deck,esfdel
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
      subroutine esfdel (ielem)
c *** primary function:    delete element surface stress data

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem    (int,sc,in)       - element number
c                                     = 0 - delete for all defined elements

c     output arguments:  none.
```

3.7.21. Function engiqr (Getting Information About an Element's Energies)

```
*deck,engiqr
      function engiqr (ielem,key)
c *** primary function: get information about element energies

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem     (int,sc,in)      - element number (or zero, see below)
c        key     (int,sc,in)        - key as to the information needed
c                 =  1 - return info about element energies
c                             ielem > 0 - return number of element energies on
c                                         this element (rec length)
c                                   = 0 - return maximum number of element
c                                         energies on any element
c                                         (max rec length)
c                 = DB_NUMDEFINED - return the number of element energies
c                                    defined in model

c     output arguments:
c        engiqr   (int,func,out)    - the returned value of engiqr is based on
c                                        setting of key
```

3.7.22. Function engget (Getting an Element's Energies)

```
*deck,engget
      function engget (ielem,value)
c *** primary function:    get element energies.

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem   (int,sc,in)           - element number

c     output arguments:
c        engget  (int,func,out)        - status of element.
c                                        =  0 - element undefined
c                                        = MAXENG+1 - energies returned (see in echprm.inc)
c        value   (dp,ar(MAXENG+1),out) - volume and energies
c                                value(1) = volume of element
c                                     (2) = stiffness energy
c                                     (3) = artificial hourglass energy
c                                     (4) = kinetic energy
c                                     (5) = plastic energy
c                                     (6) = creep energy
c                                     (7) = stabilization energy
c                                     (8) = strain energy density
c                                     (9) = thermal energy
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                    (10) = viscous regularization energy for CZM
c                                    (11) = sparse (future friction energy)
c                                    (12) = damping energy
c                                    (13) = external work by element load
c                                    (14) = stiffness energy amplitude
c                                    (15) = kinetic energy amplitude
c                                    (16) = stiffness energy peak
c                                    (17) = kinetic energy peak
c                                    (18) = intermediate result for stif. energ. amplitude
c                                           in harmonic analysis 1
c                                    (19) = intermediate result for kin. energ. amplitude
c                                           in harmonic analysis 1
c                                    (20) = intermediate result for stif. energ. amplitude
c                                           in harmonic analysis 2
c                         (21 = MAXENG+1) = intermediate result for kin. energ. amplitude
c                                           in harmonic analysis 2
```

3.7.23. Subroutine engput (Storing an Element's Energies and Volume)

```
*deck,engput
      subroutine engput (ielem,nval,value)
c *** primary function:    store volume and energies for an element.

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem  (int,sc,in)          - element number
c        nval   (int,sc,in)          - the total number of values to be stored
c                                      must be MAXENG (see in echprm.inc)
c        value  (dp,ar(MAXENG+1),in) - volume and energies
c                                value(1) = volume of element
c                                     (2) = stiffness energy
c                                     (3) = artificial hourglass energy
c                                     (4) = kinetic energy
c                                     (5) = plastic energy
c                                     (6) = creep energy
c                                     (7) = stabilization energy
c                                     (8) = strain energy density
c                                     (9) = thermal energy
c                                    (10) = viscous regularization energy for CZM
c                                    (11) = sparse (future friction energy)
c                                    (12) = damping energy
c                                    (13) = external work by element load
c                                    (14) = stiffness energy amplitude
c                                    (15) = kinetic energy amplitude
c                                    (16) = stiffness energy peak
c                                    (17) = kinetic energy peak
c                                    (18) = intermediate result for stif. energ. amplitude
c                                           in harmonic analysis 1
c                                    (19) = intermediate result for kin. energ. amplitude
c                                           in harmonic analysis 1
c                                    (20) = intermediate result for stif. energ. amplitude
c                                           in harmonic analysis 2
c                         (21 = MAXENG+1) = intermediate result for kin. energ. amplitude
c                                           in harmonic analysis 2

c     output arguments:  none
```

3.7.24. Subroutine engdel (Deleting an Element's Energies)

```
*deck,engdel
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

```
      subroutine engdel (ielem)
c *** primary function:    delete element energies

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem    (int,sc,in)       - element number
c                                     = 0 - delete for all defined elements

c     output arguments:  none.
```

3.7.25. Function egriqr (Getting Information About an Element's Nodal Gradients)

```
*deck,egriqr
      function egriqr (ielem,key)
c *** primary function: get information about element nodal gradients

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem     (int,sc,in)      - element number (or zero, see below)
c        key     (int,sc,in)        - key as to the information needed
c                 =  1 - return info about nodal gradients
c                         for ielem > 0 - return number of nodal gradients on
c                                         this element (record length)
c                                     = 0 - return maximum number of nodal
c                                         gradients on any element
c                                         (maximum record length)
c                = DB_NUMDEFINED - return the number of nodal gradients defined
c                                   in model

c     output arguments:
c        egriqr   (int,func,out)    - the returned value of egriqr is based on
c                                        setting of key
```

3.7.26. Function egrget (Getting an Element's Nodal Gradients)

```
*deck,egrget
      function egrget (ielem,value)
c *** primary function:    get element nodal gradients.

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem    (int,sc,in)       - element number

c     output arguments:
c        egrget   (int,func,out)    - status of element.
c                                     = 0 - element undefined
c                                     > 0 - number of nodal gradients
c                                           returned
c        value    (dp,ar(*),out)    - element nodal gradients

c                                    Note: If a coupled field, a set of
c                                      gradients are stored in the following
c                                      order (as available): fluid, thermal,
c                                      electric, magnetic

c *** mpg egrget < pagend,magget<hsnget2: get elem gradient, H,
```

3.7.27. Subroutine egrput (Storing an Element's Nodal Gradients)

```
*deck,egrput
      subroutine egrput (ielem,nval,value)
c *** primary function:    store nodal gradients at an element.

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem     (int,sc,in)       - element number
c        nval      (int,sc,in)       - the total number of values
c                                       (ndir*nnod*nscalr)
c        value    (dp,ar(nval),in)   - the gradient values

c                                     Note: If a coupled field, a set of
c                                       gradients are stored in the following
c                                       order (as appropriate): fluid, thermal,
c                                       electric, magnetic
c     output arguments:  none
```

3.7.28. Subroutine egrdel (Deleting an Element's Nodal Gradients)

```
*deck,egrdel
      subroutine egrdel (ielem)
c *** primary function:    delete element nodal gradients

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem    (int,sc,in)       - element number
c                                       = 0 - delete for all defined elements

c     output arguments:  none.
```

3.7.29. Function eeliqr (Getting Information About an Element's Nodal Elastic Strains)

```
*deck,eeliqr
      function eeliqr (ielem,key)
c *** primary function: get information about element nodal elastic strains

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem     (int,sc,in)      - element number (or zero, see below)
c        key     (int,sc,in)        - key as to the information needed
c                 =  1 - return info about elastic strains
c                             ielem > 0 - return number of nodal elasic strains
c                                         on this element (rec length)
c                                   = 0 - return maximum number of nodal elastic
c                                         strains on any element
c                                         (max rec length)
c                 = DB_NUMDEFINED - return the number of nodal elastic strains
c                           defined in model
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c     output arguments:
c        eeliqr   (int,func,out)    - the returned value of eeliqr is based on
c                                        setting of key
```

3.7.30. Function eelget (Getting an Element's Nodal Elastic Strains)

```
*deck,eelget
      function eelget (ielem,value)
c *** primary function:    get element nodal elastic strains.

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem    (int,sc,in)       - element number

c     output arguments:
c        eelget   (int,func,out)    - status of element.
c                                     = 0 - element undefined
c                                     > 0 - number of nodal elastic strains
c                                           returned
c        value    (dp,ar(*),out)    - element nodal elastic strains


c                                NOTE: Strains at each corner node in the order
c                                         X, Y, Z, XY, YZ, XZ, EQV
c                                      For solid elements, strains at each
c                                         corner node
c                                      For shell elements, strains at each
c                                         corner node (first top durface, then
c                                         bottom)
c                                      For layered elements (w/KEYOPT(8)=0),
c                                         strains for "first" layer at each
c                                         corner node (first at the bottom
c                                         surface of the bottom layer, then the
c                                         top surface of the top layer).
c                                         Strains for "second" layer at each
c                                         corner node (first the bottom surface,
c                                         then the top surface for the layer with
c                                         the largest failure criteria).
c                                         The second layer is not present if
c                                         failure criteria were not used or are
c                                         not appropriate
c                                      For layered elements (w/KEYOPT(8)=1),
c                                         strains for each layer at each corner
c                                         node (first at the bottom surface, then
c                                         the top surface)
c                                      For beam elements, the contents of this
c                                         record is element depenent.  See LEPEL
c                                         item of ETABLE command.
```

3.7.31. Subroutine eelput (Storing an Element's Nodal Elastic Strains)

```
*deck,eelput
      subroutine eelput (ielem,nval,value)
c *** primary function:    store nodal elastic strains at an element.

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem     (int,sc,in)       - element number
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c        nval      (int,sc,in)       - the total number of values
c                                      (7*nnod*nface)
c        value    (dp,ar(nval),in)   - nval strain values

c     output arguments:  none

c                                NOTE: Strains at each corner node in the order
c                                         X, Y, Z, XY, YZ, XZ, EQV
c                                      For solid elements, strains at each
c                                         corner node
c                                      For shell elements, strains at each
c                                         corner node (first top durface, then
c                                         bottom)
c                                      For layered elements (w/KEYOPT(8)=0),
c                                         strains for "first" layer at each
c                                         corner node (first at the bottom
c                                         surface of the bottom layer, then the
c                                         top surface of the top layer).
c                                         Strains for "second" layer at each
c                                         corner node (first the bottom surface,
c                                         then the top surface for the layer with
c                                         the largest failure criteria).
c                                         The second layer is not present if
c                                         failure criteria were not used or are
c                                         not appropriate
c                                      For layered elements (w/KEYOPT(8)=1),
c                                         strains for each layer at each corner
c                                         node (first at the bottom surface, then
c                                         the top surface)
c                                      For beam elements, the contents of this
c                                         record is element depenent.  See LEPEL
c                                         item of ETABLE command.
```

3.7.32. Subroutine eeldel (Deleting an Element's Nodal Elastic Strains)

```
*deck,eeldel
      subroutine eeldel (ielem)
c *** primary function:    delete element elastic strains

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem    (int,sc,in)       - element number
c                                      = 0 - delete for all defined elements

c     output arguments:  none.
```

3.7.33. Function epliqr (Getting Information About an Element's Nodal Plastic Strains)

```
*deck,epliqr
      function epliqr (ielem,key)
c *** primary function: get information about element nodal plastic strains

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem     (int,sc,in)      - element number (or zero, see below)
c        key     (int,sc,in)        - key as to the information needed
c                 =  1 - return info about plastic strains
c                              ielem > 0 - return number of nodal plastic strains
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                          on this element
c                                          (record length)
c                                   = 0 - return maximum number of nodal plastic
c                                          strains on any element
c                                          (max record length)
c                 = DB_NUMDEFINED - return the number of nodal plastic strains
c                                    defined in model

c     output arguments:
c        epliqr   (int,func,out)    - the returned value of epliqr is based on
c                                        setting of key
```

3.7.34. Function eplget (Getting an Element's Nodal Plastic Strains)

```
*deck,eplget
      function eplget (ielem,value)
c *** primary function:    get element nodal plastic strains.

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem    (int,sc,in)       - element number

c     output arguments:
c        eplget   (int,func,out)    - status of element.
c                                     = 0 - element undefined
c                                     > 0 - number of nodal plastic strains
c                                           returned
c        value    (dp,ar(*),out)    - element nodal plastic strains

c                                 NOTE: Strains at each corner node in the order
c                                          X, Y, Z, XY, YZ, XZ, EQV
c                                       For solid elements, strains at each
c                                          corner node
c                                       For shell elements, strains at each
c                                          corner node (first top durface, then
c                                          bottom)
c                                       For layered elements (w/KEYOPT(8)=0),
c                                          strains for "first" layer at each
c                                          corner node (first at the bottom
c                                          surface of the bottom layer, then the
c                                          top surface of the top layer).
c                                          Strains for "second" layer at each
c                                          corner node (first the bottom surface,
c                                          then the top surface for the layer with
c                                          the largest failure criteria).
c                                          The second layer is not present if
c                                          failure criteria were not used or are
c                                          not appropriate
c                                       For layered elements (w/KEYOPT(8)=1),
c                                          strains for each layer at each corner
c                                          node (first at the bottom surface, then
c                                          the top surface)
c                                       For beam elements, the contents of this
c                                          record is element depenent.  See LEPPL
c                                          item of ETABLE command.
```

3.7.35. Subroutine eplput (Storing an Element's Nodal Plastic Strains)

```
*deck,eplput
      subroutine eplput (ielem,nval,value)
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c *** primary function:    store nodal plastic strains at a element.

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem     (int,sc,in)       - element number
c        nval      (int,sc,in)       - the total number of values
c                                       (6*nnod*nface)
c        value    (dp,ar(nval),in)   - the strain values

c     output arguments:  none
c                                NOTE: Strains at each corner node in the order
c                                         X, Y, Z, XY, YZ, XZ, EQV
c                                      For solid elements, strains at each
c                                         corner node
c                                      For shell elements, strains at each
c                                         corner node (first top durface, then
c                                         bottom)
c                                      For layered elements (w/KEYOPT(8)=0),
c                                         strains for "first" layer at each
c                                         corner node (first at the bottom
c                                         surface of the bottom layer, then the
c                                         top surface of the top layer).
c                                         Strains for "second" layer at each
c                                         corner node (first the bottom surface,
c                                         then the top surface for the layer with
c                                         the largest failure criteria).
c                                         The second layer is not present if
c                                         failure criteria were not used or are
c                                         not appropriate
c                                      For layered elements (w/KEYOPT(8)=1),
c                                         strains for each layer at each corner
c                                         node (first at the bottom surface, then
c                                         the top surface)
c                                      For beam elements, the contents of this
c                                         record is element depenent.  See LEPPL
c                                         item of ETABLE command.
```

3.7.36. Subroutine epldel (Deleting an Element's Nodal Plastic Strains)

```
*deck,epldel
      subroutine epldel (ielem)
c *** primary function:    delete element plastic strains

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem    (int,sc,in)       - element number
c                                       = 0 - delete for all defined elements

c     output arguments:  none.
```

3.7.37. Function ecriqr (Getting Information About an Element's Nodal Creep Strains)

```
*deck,ecriqr
      function ecriqr (ielem,key)
c *** primary function: get information about element nodal creep strains

c *** Notice - This file contains ANSYS Confidential information ***
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c     input arguments:
c        ielem     (int,sc,in)      - element number (or zero, see below)
c        key     (int,sc,in)        - key as to the information needed
c                 =  1 - return info about creep strains
c                             ielem > 0 - return number of nodal creep strains
c                                         on this element
c                                         (record length)
c                                   = 0 - return maximum number of nodal creep
c                                         strains on any element
c                                         (max record length)
c                 = DB_NUMDEFINED - return the number of nodal creep strains
c                                    defined in model

c     output arguments:
c        ecriqr   (int,func,out)    - the returned value of ecriqr is based on
c                                        setting of key
```

3.7.38. Function ecrget (Getting an Element's Nodal Creep Strains)

```
*deck,ecrget
      function ecrget (ielem,value)
c *** primary function:    get element nodal creep strains.

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem    (int,sc,in)       - element number

c     output arguments:
c        ecrget   (int,func,out)    - status of element.
c                                     = 0 - element undefined
c                                     > 0 - number of nodal creep strains
c                                           returned
c        value    (dp,ar(*),out)    - element nodal creep strains

c                                NOTE: Strains at each corner node in the order
c                                         X, Y, Z, XY, YZ, XZ, EQV
c                                      For solid elements, strains at each
c                                         corner node
c                                      For shell elements, strains at each
c                                         corner node (first top durface, then
c                                         bottom)
c                                      For layered elements (w/KEYOPT(8)=0),
c                                         strains for "first" layer at each
c                                         corner node (first at the bottom
c                                         surface of the bottom layer, then the
c                                         top surface of the top layer).
c                                         Strains for "second" layer at each
c                                         corner node (first the bottom surface,
c                                         then the top surface for the layer with
c                                         the largest failure criteria).
c                                         The second layer is not present if
c                                         failure criteria were not used or are
c                                         not appropriate
c                                      For layered elements (w/KEYOPT(8)=1),
c                                         strains for each layer at each corner
c                                         node (first at the bottom surface, then
c                                         the top surface)
c                                      For beam elements, the contents of this
c                                         record is element depenent.  See LEPCR
c                                         item of ETABLE command.
```

3.7.39. Subroutine ecrput (Storing an Element's Nodal Creep Strains)

```
*deck,ecrput
      subroutine ecrput (ielem,nval,value)
c *** primary function:    store nodal creep strains at an element.

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem     (int,sc,in)       - element number
c        nval      (int,sc,in)       - the total number of values
c                                       (6*nnod*nface)
c        value    (dp,ar(nval),in)   - the strain values

c     output arguments:  none
c                               NOTE: Strains at each corner node in the order
c                                        X, Y, Z, XY, YZ, XZ, EQV
c                                     For solid elements, strains at each
c                                        corner node
c                                     For shell elements, strains at each
c                                        corner node (first top durface, then
c                                        bottom)
c                                     For layered elements (w/KEYOPT(8)=0),
c                                        strains for "first" layer at each
c                                        corner node (first at the bottom
c                                        surface of the bottom layer, then the
c                                        top surface of the top layer).
c                                        Strains for "second" layer at each
c                                        corner node (first the bottom surface,
c                                        then the top surface for the layer with
c                                        the largest failure criteria).
c                                        The second layer is not present if
c                                        failure criteria were not used or are
c                                        not appropriate
c                                     For layered elements (w/KEYOPT(8)=1),
c                                        strains for each layer at each corner
c                                        node (first at the bottom surface, then
c                                        the top surface)
c                                     For beam elements, the contents of this
c                                        record is element depenent.  See LEPCR
c                                        item of ETABLE command.
```

3.7.40. Subroutine ecrdel (Deleting an Element's Nodal Creep Strains)

```
*deck,ecrdel
      subroutine ecrdel (ielem)
c *** primary function:    delete element creep strains

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem    (int,sc,in)       - element number
c                                       = 0 - delete for all defined elements

c     output arguments:  none.
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.7.41. Function ethiqr (Getting Information About an Element's Nodal Thermal Strains)

```
*deck,ethiqr
      function ethiqr (ielem,key)
c *** primary function: get information about element nodal thermal strains

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem     (int,sc,in)      - element number (or zero, see below)
c        key     (int,sc,in)        - key as to the information needed
c                 =  1 - return info about thermal strains
c                             ielem > 0 - return number of nodal thermal strains
c                                         on this element
c                                         (record length)
c                                   = 0 - return maximum number of nodal thermal
c                                         strains on any element
c                                         (max record length)
c                 = DB_NUMDEFINED - return the number of nodal thermal strains
c                                    defined in model

c     output arguments:
c        ethiqr   (int,sc,out)      - the returned value of ethiqr is based on
c                                        setting of key
```

3.7.42. Function ethget (Getting an Element's Nodal Thermal Strains)

```
*deck,ethget
      function ethget (ielem,value)
c *** primary function:    get element nodal thermal strains.
c                            also the volumetric swelling strain
c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem    (int,sc,in)       - element number

c     output arguments:
c        ethget   (int,func,out)    - status of element.
c                                     = 0 - element undefined
c                                     > 0 - number of nodal thermal strains
c                                           returned
c        value    (dp,ar(*),out)    - element nodal thermal strains

c                                NOTE: Strains at each corner node in the order
c                                         X, Y, Z, XY, YZ, XZ, EQV, epswel
c                                      For solid elements, strains at each
c                                         corner node
c                                      For shell elements, strains at each
c                                         corner node (first top durface, then
c                                         bottom)
c                                      For layered elements (w/KEYOPT(8)=0),
c                                         strains for "first" layer at each
c                                         corner node (first at the bottom
c                                         surface of the bottom layer, then the
c                                         top surface of the top layer).
c                                         Strains for "second" layer at each
c                                         corner node (first the bottom surface,
c                                         then the top surface for the layer with
c                                         the largest failure criteria).
c                                         The second layer is not present if
c                                         failure criteria were not used or are
c                                         not appropriate
c                                      For layered elements (w/KEYOPT(8)=1),
```

```
c                                       strains for each layer at each corner
c                                       node (first at the bottom surface, then
c                                       the top surface)
c                                    For beam elements, the contents of this
c                                       record is element depenent.  See LEPTH
c                                       item of ETABLE command.
```

3.7.43. Subroutine ethput (Storing an Element's Nodal Thermal Strains)

```
*deck,ethput
      subroutine ethput (ielem,nval,value)
c *** primary function:    store nodal thermal strains at an element.
c                           also the volumetric swelling strain

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem     (int,sc,in)       - element number
c        nval      (int,sc,in)       - the total number of values
c                                       (7*nnod*nface)
c        value    (dp,ar(nval),in)   - the strain values

c     output arguments:  none
c                               NOTE: Strains at each corner node in the order
c                                       X, Y, Z, XY, YZ, XZ, EQV, epswel
c                                     For solid elements, strains at each
c                                       corner node
c                                     For shell elements, strains at each
c                                       corner node (first top durface, then
c                                       bottom)
c                                     For layered elements (w/KEYOPT(8)=0),
c                                       strains for "first" layer at each
c                                       corner node (first at the bottom
c                                       surface of the bottom layer, then the
c                                       top surface of the top layer).
c                                       Strains for "second" layer at each
c                                       corner node (first the bottom surface,
c                                       then the top surface for the layer with
c                                       the largest failure criteria).
c                                       The second layer is not present if
c                                       failure criteria were not used or are
c                                       not appropriate
c                                     For layered elements (w/KEYOPT(8)=1),
c                                       strains for each layer at each corner
c                                       node (first at the bottom surface, then
c                                       the top surface)
c                                     For beam elements, the contents of this
c                                       record is element depenent.  See LEPTH
c                                       item of ETABLE command.
```

3.7.44. Subroutine ethdel (Deleting an Element's Thermal, Initial, and Swelling Strains)

```
*deck,ethdel
      subroutine ethdel (ielem)
c *** primary function:    delete element thermal, initial, and
c                          swelling strains

c *** Notice - This file contains ANSYS Confidential information ***
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c     input arguments:
c        ielem    (int,sc,in)       - element number
c                                     = 0 - delete for all defined elements

c     output arguments:  none.
```

3.7.45. Function euliqr (Getting Information About an Element's Euler Angles)

```
*deck,euliqr
      function euliqr (ielem,key)
c *** primary function: get information about element euler angles

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem     (int,sc,in)      - element number (or zero, see below)
c        key       (int,sc,in)      - key as to the information needed
c                 =  1 - return info about element euler angles
c                             ielem > 0 - return number of euler angles on this
c                                         element
c                                         (record length)
c                                   = 0 - return maximum number of euler angles
c                                         on any element
c                                         (max record length)
c                 = DB_NUMDEFINED - return the number of element euler angles
c                                    defined in model

c     output arguments:
c        euliqr   (int,func,out)    - the returned value of euliqr is based on
c                                        setting of key
```

3.7.46. Function eulget (Getting an Element's Nodal Euler Angles)

```
*deck,eulget
      function eulget (ielem,value)
c *** primary function:    get element nodal euler angles.

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem    (int,sc,in)       - element number

c     output arguments:
c        eulget   (int,func,out)    - status of element.
c                                     = 0 - element undefined
c                                     > 0 - number of euler angle values
c                                           returned
c        value    (dp,ar(*),out)    - element euler angles

c                                  NOTE: For lower-ordered elements, rotations
c                                          at centroid
c                                        For higher-ordered elements, rotations
c                                          at each corner node
c                                        For layered shells, rotations at each
c                                          corner node, plus layer rotation angle
c                                          for each layer (real constant THETA)
c                                        For layered solids, rotation angles at
c                                          centroid, plus layer rotation angle
c                                          for each layer (real constant THETA)
c                                        For surface element, no euler angles
c                                          are saved
```

3.7.47. Subroutine eulput (Storing an Element's Euler Angles)

```
*deck,eulput
      subroutine eulput (ielem,nval,value)
c *** primary function:    store nodal euler angles for an element.

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem     (int,sc,in)       - element number
c        nval      (int,sc,in)       - the total number of values
c                                       (3 * number of display nodes)
c        value    (dp,ar(nval),in)   - the euler angle values

c     output arguments:  none
c                                NOTE: For lower-ordered elements, rotations
c                                        at centroid
c                                      For higher-ordered elements, rotations
c                                        at each corner node
c                                      For layered shells, rotations at each
c                                        corner node, plus layer rotation angle
c                                        for each layer (real constant THETA)
c                                      For layered solids, rotation angles at
c                                        centroid, plus layer rotation angle
c                                        for each layer (real constant THETA)
```

3.7.48. Subroutine euldel (Deleting an Element's Euler Angles)

```
*deck,euldel
      subroutine euldel (ielem)
c *** primary function:    delete element euler angles

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem    (int,sc,in)       - element number
c                                       = 0 - delete for all defined elements

c     output arguments:  none.
```

3.7.49. Function efxiqr (Getting Information About Element Fluxes)

```
*deck,efxiqr
      function efxiqr (ielem,key)
c *** primary function:    get information about element fluxes

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem     (int,sc,in)      - element number (or zero, see below)
c        key       (int,sc,in)      - key as to the information needed
c                 =  1 - return info about element fluxes
c                             ielem > 0 - return number of fluxes on this
c                                         element
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                          (record length)
c                                  = 0 - return maximum number of fluxes
c                                          on any element
c                                          (max record length)
c                 = DB_NUMDEFINED - return the number of element fluxes defined
c                                    in model

c     output arguments:
c        efxiqr   (int,func,out)    - the returned value of efxiqr is based on
c                                     setting of key
```

3.7.50. Function efxget (Getting an Element Flux)

```
*deck,efxget
      function efxget (ielem,value)
c *** primary function:    get element nodal fluxes.

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem    (int,sc,in)       - element number

c     output arguments:
c        efxget   (int,func,out)    - status of element.
c                                     = 0 - element undefined
c                                     > 0 - number of nodal fluxes returned
c        value    (dp,ar(*),out)    - element nodal fluxes

c                                  Note: If a coupled field, a set of fluxes is
c                                        stored in the following order (as
c                                        available):  fluid, thermal,
c                                        electric, magnetic

c *** mpg efxget<pagend<paberrwb,edgzzx,panavg,papres,paterr: get ele nd flx, B
```

3.7.51. Subroutine efxput (Storing an Element's Fluxes)

```
*deck,efxput
      subroutine efxput (ielem,nval,value)
c *** primary function:    store nodal fluxes at an element.

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem     (int,sc,in)       - element number
c        nval      (int,sc,in)       - the total number of values
c                                      (ndir*nnod*nscalr)
c        value    (dp,ar(nval),in)   - the flux values

c     output arguments:  none
c                                  Note: If a coupled field, a set of fluxes is
c                                        stored in the following order (as
c                                        available):  fluid, thermal,
c                                        electric, magnetic
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.7.52. Subroutine efxdel (Deleting Element Fluxes)

```
*deck,efxdel
      subroutine efxdel (ielem)
c *** primary function:    delete element nodal fluxes

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem    (int,sc,in)       - element number
c                                     = 0 - delete for all defined elements

c     output arguments:  none.
```

3.7.53. Function elfiqr (Getting Information About Element Local Forces)

```
*deck,elfiqr
      function elfiqr (ielem,key)
c *** primary function: get information about elem local forces

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem     (int,sc,in)      - element number (or zero, see below)
c        key       (int,sc,in)      - key as to the information needed
c                 =  1 - return info about element local forces
c                             ielem > 0 - return number of local forces on this
c                                         element
c                                         (record length)
c                                   = 0 - return maximum number of local forces
c                                         on any element
c                                         (max record length)
c                 = DB_NUMDEFINED - return the number of element local forces
c                                    defined in model

c     output arguments:
c        elfiqr   (int,func,out)    - the returned value of elfiqr is based on
c                                        setting of key
```

3.7.54. Function elfget (Getting an Element Local Force)

```
*deck,elfget
      function elfget (ielem,value)
c *** primary function:    get element local nodal forces.

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem    (int,sc,in)       - element number

c     output arguments:
c        elfget   (int,func,out)    - status of element.
c                                     = 0 - element has no local nodal forces
c                                     > 0 - number of nodal forces returned
c        value    (dp,ar(*),out)    - element local nodal forces.

c *** mpg elfget<pagend<paberrwb,edgzzx,panavg,papres,paterr: get ele nd frc, F
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.7.55. Subroutine elfput (Storing an Element's Local Forces)

```
*deck,elfput
      subroutine elfput (ielem,nval,value)
c *** primary function:    store element local nodal forces.

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c     variable (typ,siz,intent)     description
c        ielem     (int,sc,in)       - element number
c        nval      (int,sc,in)       - the total number of values
c                                       NOTE: There may be a maximum of 3 sets of
c                                      nodal forces in the record: static
c                                      forces, inertia forces, and damping forces
c        value    (dp,ar(nval),in)   - element local nodal forces

c     output arguments:  none
```

3.7.56. Subroutine elfdel (Deleting Element Local Forces)

```
*deck,elfdel
      subroutine elfdel (ielem)
c *** primary function:    delete element local forces

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem    (int,sc,in)       - element number
c                                       = 0 - delete for all defined elements

c     output arguments:  none.
```

3.7.57. Function emniqr (Getting Information About Element Miscellaneous Non-summable Data)

```
*deck,emniqr
      function emniqr (ielem,key)
c *** primary function:    get information about element misc non-summable
c                          data

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem     (int,sc,in)      - element number (or zero, see below)
c        key       (int,sc,in)      - key as to the information needed
c                 =  1 - return info about element misc non-summed data
c                             ielem > 0 - return number of data items on this
c                                         element
c                                         (record length)
c                                    = 0 - return maximum number of data items
c                                         on any element
c                                         (max record length)
c                 = DB_NUMDEFINED - return the number of element misc non-summed
c                                    data items defined in model

c     output arguments:
c        emniqr   (int,func,out)    - the returned value of emniqr is based on
```

```
c                                       setting of key
```

3.7.58. Function emnget (Getting an Element's Miscellaneous Non-summable Data)

```
*deck,emnget
      function emnget (ielem,value)
c *** primary function:    get misc non-summable data.

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem    (int,sc,in)       - element number

c     output arguments:
c        emnget   (int,func,out)    - status of element.
c                                     = 0 - no non-summed misc data at this
c                                           element
c                                     > 0 - number of data items returned
c        value    (dp,ar(*),out)    - element misc non-summed data.

c                                    NOTE: the contents of this record is element
c                                        dependent.  See NMISC on ETABLE command
```

3.7.59. Subroutine emnput (Storing an Element's Miscellaneous Non-summable Data)

```
*deck,emnput
      subroutine emnput (ielem,nval,value)
c *** primary function:    store misc. non-summable data for an element.

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem     (int,sc,in)       - element number
c        nval      (int,sc,in)       - the total number of values
c        value    (dp,ar(nval),in)   - the misc. non-summed data items

c     output arguments:  none
c                                    NOTE: the contents of this record is element
c                                        dependent.  See NMISC on ETABLE command
```

3.7.60. Subroutine emndel (Deleting an Element's Miscellaneous Non-summable Data)

```
*deck,emndel
      subroutine emndel (ielem)
c *** primary function:    delete element misc non-summable data

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem    (int,sc,in)       - element number
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                     = 0 - delete for all defined elements

c     output arguments:  none.
```

3.7.61. Function ecdiqr (Getting Information About Element Current Densities)

```
*deck,ecdiqr
      function ecdiqr (ielem,key)
c *** primary function:    get information about element current densities

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem     (int,sc,in)      - element number (or zero, see below)
c        key       (int,sc,in)      - key as to the information needed
c                 =  1 - return info about element current densities
c                             ielem > 0 - return number of current densities on
c                                         this element
c                                         (record length)
c                                   = 0 - return maximum number of current
c                                         densities on any element
c                                         (max record length)
c                 = DB_NUMDEFINED - return the number of element current
c                                    densities defined in model

c     output arguments:
c        ecdiqr   (int,func,out)    - the returned value of ecdiqr is based on
c                                        setting of key
```

3.7.62. Function ecdget (Getting an Element Current Density)

```
*deck,ecdget
      function ecdget (ielem,value)
c *** primary function:    get calculated element current densities.

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem    (int,sc,in)       - element number

c     output arguments:
c        ecdget   (int,func,out)    - status of element.
c                                     = 0 - element has no current densities
c                                     > 0 - number of calculated element
c                                           current densities
c        value    (dp,ar(*),out)    - calculated element current densities.

c                                    NOTE: current densities are in the order
c                                          X, Y, Z
```

3.7.63. Subroutine ecdput (Storing an Element's Current Densities)

```
*deck,ecdput
      subroutine ecdput (ielem,nval,value)
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c *** primary function:    store calculated element current densities

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem    (int,sc,in)        - element number
c        nval     (int,sc,in)        - the total number of values
c        value    (dp,ar(nval),in)   - calculated element current densities.

c     output arguments:   none
c                                  NOTE: current densities are in the order
c                                        X, Y, Z
```

3.7.64. Subroutine ecddel (Deleting Element Current Densities)

```
*deck,ecddel
      subroutine ecddel (ielem)
c *** primary function:    delete element current densities

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem    (int,sc,in)       - element number
c                                     = 0 - delete for all defined elements

c     output arguments:  none.
```

3.7.65. Function enliqr (Getting Information About Element Nonlinear Tables)

```
*deck,enliqr
      function enliqr (ielem,key)
c *** primary function: get information about element nonlinear tables

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem     (int,sc,in)      - element number (or zero, see below)
c        key       (int,sc,in)      - key as to the information needed
c                =  1 - return info about element nonlinear tables
c                            ielem > 0 - return number of nonlinear tables for
c                                        this element
c                                        (record length)
c                                   = 0 - return maximum number of nonlinear
c                                        tables for any element
c                                        (max record length)
c                = DB_NUMDEFINED - return the number of element nonlinear
c                                   tables defined in model

c     output arguments:
c        enliqr   (int,func,out)    - the returned value of enliqr is based on
c                                     setting of key
```

3.7.66. Function enlget (Getting Element Nonlinear Tables)

```
*deck,enlget
      function enlget (ielem,value)
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c *** primary function:    get element nonlinear tables.

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem    (int,sc,in)       - element number

c     output arguments:
c        enlget   (int,func,out)    - status of element.
c                                     = 0 - nonlinear tables undefined
c                                     > 0 - number of nonlinear tables defined
c        value    (dp ,ar(n),out)   - the element nonlinear tables.

c                                    NOTE: Nonlinear data at each node are in the
c                                           order SEPL, SRAT, HPRES, EPEQ, PSV,
c                                           PLWK, and 4 spares
c                                          For beam elements, the contents and
c                                           number of information is element
c                                           dependent.  See NLIN on ETABLE
c                                           command
```

3.7.67. Subroutine enlput (Storing an Element's Nonlinear Tables)

```
*deck,enlput
      subroutine enlput (ielem,n,temp)
c *** primary function:    store element nonlinear tables

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem    (int,sc,in)       - element number
c        n        (int,sc,in)       - number of element nonlinear table values
c        temp     (dp ,ar(6),in)    - element nonlinear table,etc.

c     output arguments:  none.
c                                    NOTE: Nonlinear data at each node are in the
c                                           order SEPL, SRAT, HPRES, EPEQ, PSV,
c                                           PLWK, and 4 spares
c                                          For beam elements, the contents and
c                                           number of information is element
c                                           dependent.  See NLIN on ETABLE
c                                           command
```

3.7.68. Subroutine enldel (Deleting Element Nonlinear Tables)

```
*deck,enldel
      subroutine enldel (ielem)
c *** primary function:    delete element nonlinear tables

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem    (int,sc,in)       - element number
c                                     = 0 - delete for all defined elements

c     output arguments:  none.
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.7.69. Function ehciqr (Getting Information About Calculated Element Heat Generations)

```
*deck,ehciqr
      function ehciqr (ielem,key)
c *** primary function: get information about calculated elem heat generations

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem     (int,sc,in)      - element number (or zero, see below)
c        key       (int,sc,in)      - key as to the information needed
c                 =  1 - return info about calculated element heat gens
c                        for ielem > 0 - return number of heat gens for
c                                        this element
c                                        (record length)
c                                  = 0 - return maximum number of heat gens
c                                        for any element
c                                        (max record length)
c                 = DB_NUMDEFINED - return the number of calculated element heat
c                                   generations defined in model

c     output arguments:
c        ehciqr   (int,func,out)    - the returned value of ehciqr is based on
c                                     setting of key
```

3.7.70. Function ehcget (Getting a Calculated Element Heat Generation)

```
*deck,ehcget
      function ehcget (ielem,value)
c *** primary function:    get calculated element heat generations.

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem    (int,sc,in)       - element number

c     output arguments:
c        ehcget   (int,func,out)    - status of element.
c                                     = 0 - element undefined
c                                     > 0 - number of calculated element
c                                           heat generations
c        value    (dp,ar(*),out)    - calculated element heat generations.
```

3.7.71. Subroutine ehcput (Storing an Element's Calculated Heat Generations)

```
*deck,ehcput
      subroutine ehcput (ielem,nval,value)
c *** primary function:    store calculated element heat generations

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem    (int,sc,in)        - element number
c        nval     (int,sc,in)        - the total number of values
c        value    (dp,ar(nval),in)   - calculated element heat generations.

c     output arguments:  none
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.7.72. Subroutine ehcdel (Deleting Element Calculated Heat Generations)

```
*deck,ehcdel
      subroutine ehcdel (ielem)
c *** primary function:    delete calculated element heat generations

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        ielem    (int,sc,in)       - element number
c                                     = 0 - delete for all defined elements

c     output arguments:  none.
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 4: Subroutines for Your Convenience

This chapter describes routines available to you for use in programming. Using these routines isn't required, but may make your life easier. These routines include a set of general routines that perform utility-type functions, a set of routines supporting vector functions, a set of routines supporting matrix functions, and routines supporting message processing options.

The following topics are discussed in this chapter:

4.1. Input and Output Abbreviations

The descriptions of inputs and outputs for the routines discussed in this chapter use the following abbreviations:

- Argument *type* is one of the following:

  int - integer
  dp - double precision
  log - logical
  chr - character
  dcp - double precision complex

- Argument *size* is one of the following:

  sc - scalar variable
  ar(*n*) - array variable of length *n*
  func - functional return value

- Argument *intent* is one of the following:

  in - input argument
  out - output argument
  inout - both an input and an output argument

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.2. General Subroutines

The following general subroutines are available for your convenience:

4.2.1. Subroutine dptoch (Retrieve Eight Characters From a Double Precision Variable)

```
*deck,dptoch
      subroutine dptoch (dp8,ch8)
c *** primary function:    retreive 8 characters from a dp variable

c *** Notice - This file contains ANSYS Confidential information ***

c !!! NOTICE to progammers: this routine does not convert from a !!!
c !!! machine-independent format! Use dpexttoch if this dp word  !!!
c !!! came from a common or non-char database record             !!!

c  input arguments:
c     dp8      (dp,sc,in)        - dp variable containing characters

c  output arguments:
c     ch8      (ch*8,sc,out)     - characters retreived from the dp word
```

4.2.2. Function ppinqr (Obtain Information About Threads)

```
*deck,ppinqr           parallel
      function ppinqr (key)

c primary function:    Get information from pplib (parallel library)

c keywords:  subroutine to inquire from parallel library

c object/library:
c         current - pplib

c *** Notice - This file contains ANSYS Confidential information ***

c  input arguments:
c     key      (int,sc,in)       -  0, Get ppNprocs
c                                       (if >1, parallel allowed)
c                                  1, Get ppOff
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c                                        (0,pp active 1, inactive)
c                                     2, Get ppMaxProc
c                                        (max processors to be used)
c                                     3, Get ppNumThreads
c                                        (number of threads in set)
c                                     4, Get ppThdRunning
c                                        (either 0 or the number of threads running)
c                                     5, Get ppDebug
c                                        (debug level value)
c                                     7, -1, parallel not started
c                                         0, parallel started, threads not
c                                         n, number of threads running
c                                    10, Get ppNprocOrig
c                                    12, Get ppInitialize
c                                    13, Get ppRunLevel
c                                    22, Get ppLockCheck
c                                    23, Get ppHybrid

c  output arguments:
c     ppinqr   (int,sc,out)      - The value of the referenced variable
```

For information about using this function, see Sharing Data Between User Routines (p. 137).

4.2.3. Function pplock (Locking a Thread in Shared Memory)

```
*deck,pplock           parallel
      subroutine pplock (ilock)
c **********  set the lock ilock **********
```

For information about using this function, see Sharing Data Between User Routines (p. 137).

4.2.4. Function ppunlock (Unlocking a Thread in Shared Memory)

```
*deck,ppunlock           parallel
      subroutine ppunlock(ilock)
c **********  clear the lock ilock **********
```

For information about using this function, see Sharing Data Between User Routines (p. 137).

4.2.5. Function ppproc (Get the Active Thread Index)

```
*deck,ppproc           parallel
      function ppproc ()
c *** primary function:    return unique thread number (0:ppNprocs-1)

c *** Notice - This file contains ANSYS Confidential information ***

c  output arguments:
c     ppproc   (int,sc,out)      - the thread number for this process
```

For information about using this function, see Sharing Data Between User Routines (p. 137).

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.2.6. Function wrinqr (Obtain Information About Output)

```
*deck,wrinqr
      function wrinqr (key)
c *** primary function:   obtain information about output

c *** Notice - This file contains ANSYS Confidential information ***

c --- caution: the following variables are "saved/resumed".
c ---          key=WR_COLINTER thru WR_SUPCOLMAX in "wrinqr/wrinfo"
c ---           (data for "/fmt,/page,/header" commands).
c ---           note that the whole common cannot be "saved/resumed".  cwa

c     typ=int,dp,log,chr,dcp   siz=sc,ar(n),func    intent=in,out,inout

c  input arguments:
c     variable (typ,siz,intent)    description                        wrcom name
c     key      (int,sc,in)
c                  = WR_PRINT       - print flag (kprint)                 prtkey
c                       wrinqr = 0 - no output
c                              = 1 - print
c                  = WR_OUTPUT      - current output unit number(iott) outfil
c                  = WR_MASTEROUT   - master output file                frstot
c                  = WR_COLINTER    - interactive columns per page      intcol
c                  = WR_COLBATCH    - batch columns per page            batcol
c                  = WR_LINEINTER   - interactive lines per page        intlin
c                  = WR_LINEBATCH   - batch lines per page              batlin
c                  = WR_COMMASEP    - 1 for comma separated output      CommaSep
c                  = WR_CHARITEM    - characters per output item        chrper
c                  = WR_CHARDECIMAL - characters past decimal           chrdec
c                  = WR_CHARINTEGER - characters in leading integer     chrint
c                  = WR_CHARTYPE    -                                   chrtyp
c                        wrinqr = 1 - using E format in output
c                               = 2 - using F format in output
c                               = 3 - using G format in output
c                  = WR_SUPTITLE    - tlabel supress key                keyhed
c                  = WR_SUPSUBTITLE - subtitle supress key              keytit
c                  = WR_SUPLSITER   - ls,iter id supress key            keyid
c                  = WR_NOTELINE    - note line supress key             keynot
c                  = WR_SUPCOLHEADER - column header supress key        keylab
c                  = WR_SUPCOLMAX   - column maximum supress key        keysum
c                  = WR_LISTOPT     - ListOpt from /output command      ListOpt

c  output arguments:
c     wrinqr   (int,func,out)      - the value corresponding to key
```

4.2.7. Subroutine erinqr (Obtaining Information from the Errors Common)

```
*deck,erinqr
      function erinqr (key)
c *** primary function:    obtain information from errors common
c
c *** Notice - This file contains ANSYS Confidential information ***
c
c  input arguments:
c     key      (int,sc,in)       - item to be returned
c                                  1=keyerr, 2=errfil,    3=numnot, 4=numwrn,
c                                  5=numerr, 6=numfat,    7=maxmsg, 8=lvlerr,
c                                  9=mxpcmd, 10=nercmd,  11=nertim,12=nomore,
c                                  13=eropen,14=ikserr,  15=kystat,16=mxr4r5,
c                                  17=mshkey,            19=opterr,20=flowrn,
c                                            22=noreport,23=pdserr,24=mxpcmdw
c                                  25=kystop,26=icloads, 27=ifkey,
c                                  28=intrupt
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c
c ---- below definitions copied from errcom 7/92 for user information
c
c                            *** key number= ........................
c                    (see ansysdef for parameter definitions)          |
c                                                                      \/
c
co keyerr - master error flag                                    (ER_ERRORFLAG)
co errfil - errors file unit number                              (ER_ERRORFILE)
co numnot - total number of notes displayed                      (ER_NUMNOTE)
co numwrn - total number of warnings displayed                   (ER_NUMWARNING)
co numerr - total number of errors displayed                     (ER_NUMERROR)
co numfat - total number of fatals displayed                     (ER_NUMFATAL)
co maxmsg - max allowed number of displayed messages before abort(ER_MAXMESSAGE)
co lvlerr - used basicly in solution (from cnvr command.)        (ER_ERRORLEVEL)
co           -1=do not set keyerr for notes/errors/warnings.
co           -2=same as -1 but do not display message either.
co mxpcmd - maximum number of messages allowed per command       (ER_MAXCOMMAND)
co nercmd - number of messages displayed for any one command     (ER_NUMCOMMAND)
co nertim - key as to how message cleared from u/i pop-up        (ER_UICLEAR)
co           (as per rsg/pft 5/1/92 - only for "info" calls
co            -1=message is timed before removal
co             0=message needs pick or keyboard before removal
co             1=message stays up untill replaced by another message
co nomore   display any more messages                            (ER_NOMOREMSG)
co            0=display messages
co            1=display discontinue message and stop displaying
co eropen - 0=errors file is closed                              (ER_FILEOPEN)
co           1=errors file is opened
co ikserr - 0=if interactive do not set keyerr                   (ER_INTERERROR)
c         - 1=if interactive set keyerr (used by mesher and tessalation)
co kystat - flag to bypass keyopt tests in the elcxx routines    (ER_KEYOPTTEST)
c           associated with status/panel info  inquiries.
c            0=do not bypass keyopt tests
c            1=perform all keyopt tests
c           also flag to bypass setting of _STATUS upon resume
co mxr4r5 - mixed rev4-rev5 input logic (*do,*if,*go,*if-go)     (ER_MIXEDREV)
c            (used in chkmix called from rdmac)
c            1=rev5 found (*do,*fi-then-*endif)
c            2=rev4 found (*go,:xxx,*if,....,:xxx)
c            3=warning printed. do not issue any more.
co mshkey - cpu intensive meshing etc. this will cause           (ER_MESHING)
c           "nertim (11)" to be set to -1 for "notes", 1 for "warnings",
c           and 0 for "errors". checking of this key is done in "anserr".
c            0=not meshing or cpu intensive
c            1=yes, meshing or cpu intensive
co syerro - systop error code. read by anserr if set.            (18)
co opterr - 0=no error in main ansys during opt looping          (ER_OPTLOOPING)
c           1=an error has happened in main ansys during opt looping
co flowrn - flag used by "floqa" as to list floqa.ans            (20)
c           0=list "floqa.ans"
c           1="floqa.ans" has been listed. do not list again.
co noreport- used in GUI for turning off errors due to strsub calls (22)
c           0=process errors as usual
c           1=do NOT report errors
co pdserr - 0=no error in main ansys during pds looping          (ER_PDSLOOPING)
c           1=an error has happened in main ansys during pds looping
co mxpcmdw- number of messages written to file.err for any one   (24)
co          command
c           0=write all errors to file.err
c           1=only write displayed errors to file.err
co icloads - key to forbid the iclist command from listing solution    (26)
c            data instead of the input data.
c           0=iclist is OK
c           1=do not permit iclist
co ifkey   - key on whether or not to abort during /input on error     (27)
c           0=do not abort
c           1=abort
co intrupt - interrupt button, so executable returns no error    (ER_INTERRUPT)
c
co espare - spare integer variables
```

```
c
c  --- end of information from errcom
c
c  output arguments:
c     erinqr   (int,sc,out)      - value corresponding to key
c
c *** mpg erinqr < el117,el115,el126,el109,el53,el96,el97,edg?: get error stat
c
```

4.2.8. Subroutine erhandler (Displaying Program Errors)

```
*deck,erhandler
      subroutine erhandler  (filein,msgid,msglvl,lngstrng,dperr,cherr)

c *** primary function:    Display ANSYS error messages

c *** Notice - This file contains ANSYS Confidential information ***

c  input arguments:
c
c               In order to make life for vismg easier,
c               do NOT use variables for the first four arguments
c
c     filein   (ch*40,sc,in)     - Filename used for character portion of
c                                  message ID (this is the file name of the
c                                  file which contains the source for this
c                                  routine)
c
c                                  if 'ErrorMessageProbe', then error was
c                                    generated on another processor (distributed
c                                    ANSYS). In that case, dperr contains the
c                                    message already made ASCII and expanded
c
c     msgid    (int,sc,in)       - Numeric portion of the message ID
c                                  1 - 9999, unique for each erhandler
c                                  call in the FILE.  Recommend using
c                                  a sequence, similar to format conventions,
c                                  i.e., 5000, 5010, 5020
c                                  if filein='ErrorMessageProbe', this is the
c                                    CPU # that originally generated the error
c     msglvl   (int,sc,in)       - level of error (same as lngerr)
c                                  0=no label (used for u/i pop-ups)
c                                 -1=no label (used for u/i pop-ups) timed
c                                    as a note message
c                                  1=note, 2=warning, 3=error, 4=fatal
c                                                   -3=error w/tech supp note
c                                                   -4=fatal w/tech supp note
c                                     (see lngerr.F for text of tech supp note)
c     lngstrng (ch*(*),sc,in)    - error message to display. use keywords
c                                  of %i %g %c %/ for formating (same as
c                                  lngerr)
c     dperr    (dp,ar(*),in)     - vector of data to display. contains both
c                                  integer and double precision data.
c                                  (same as lngerr)
c                                    if filein='ErrorMessageProbe', dperr
c                                    contains the unpacked message and lngstrng
c                                    and cherr are ignored
c     cherr    (ch*(*),ar(*),in) - vector of character data to display
c                                  max length of character data is 32
c                                  characters
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.2.9. Subroutine intrp (Doing Single Interpolation)

```
*deck,intrp
      subroutine intrp (klog,kppx,kstpz,xval,ax,ay,yval,nmax,kyoff)
c *** primary function: **** subroutine for single interpolation ****
c                       (if double interpolation is needed, see intrpt)
c
c *** Notice - This file contains ANSYS Confidential information ***
c
c     typ=int,dp,log,chr,dcp   siz=sc,ar(n),func    intent=in,out,inout
c
c  input arguments:
c  variable (typ,siz,intent)      description
c     klog      (int,sc,in)       - interpolation type
c                                   = 0 - use linear interpolation
c                                   = 1 - use log-log interpolation
c                                     -- note:  there is no option yet for
c                                               lin-log or log-lin
c     kppx      (int,sc,in)       - X value end of table signal
c                                   = 0 - a repeated x-value will signal the end
c                                         of the table
c                                   = 1 - a repeated x-value will not signal the
c                                         end of the table
c                                         (only known use = c evaluation)
c     kstpz     (int,sc,in)       - Y value end of table signal
c                                   = 0 - a yval of zero will not signal the end
c                                         of the table (e.g. stress fitting)
c                                   = 1 - a yval of zero will signal the end of
c                                         the table (in general, material
c                                         properties (exception: alpx))
c
c                          NOTE: the end of the table will be signaled thru
c                                either of the above conditions, or more
c                                commonly, that nmax values have been processed,
c                                or that the present x table entry is less than
c                                the previous one (ax(i) .lt. ax(i-1)).
c                                evaluations done after the end of the table are
c                                evaluated as if they were at the end of the
c                                table. similarly, evaluations done before the
c                                beginning of the table are done as if they were
c                                done at the beginning of the table.
c
c     xval     (dp,sc,in)         - value of x with which to go into the table
c     ax       (dp,ar(*),in)      - table of x values, in ascending order
c     ay       (dp,ar(*),in)      - table of y values
c     nmax     (int,sc,in)        - maximum table size allowed
c
c  output arguments:
c     yval     (dp,sc,out)        - value of y which comes back from the table
c     kyoff    (int,sc,out)       - xval status flag
c                                   = 0 - xval in x range
c                                   = 1 - xval less than minimum x
c                                   = 2 - xval greater than maximum x
c
```

4.2.10. Subroutine tranx3 (Processing Geometry for 3-D Line Elements)

```
*deck,tranx3
      subroutine tranx3 (nnod,xyz,nx,tr)
c *** primary function: geometric processor for 3-d line elements
c                       with or without a 3rd node
c
c *** Notice - This file contains ANSYS Confidential information ***
c
c  input arguments:
```

```
c     nnod     (int,sc,in)       - number of nodes (2 or 3)
c     xyz      (dp,ar(nx,*),in)  - coordinates (x,y,z down)
c     nx       (int,sc,in)       - row dimension of xyz array
c
c  output arguments:
c     tr       (dp,ar(3,3),in)   - transformation matrix
c
```

4.2.11. Subroutine systop (Stopping a Program Run)

```
*deck,systop
      subroutine systop (icode)
c *** primary function:    stop an ansys run
c *** secondary functions: pass an error code to the system
c
c *** Notice - This file contains ANSYS Confidential information ***
c
c  input arguments:
c     icode    (int,sc,in)       - stop error code (0<icode<127)
c                                    0 - normal exit
c                                    1 - stack overflow error
c                                    2 - stack level overflow
c                                    3 - stack pop below zero
c                                    4 - names do not match in stkpxp
c                                    5 - command line argument error
c                                    6 - unused (was: accounting file error)
c                                    7 - licensing failure
c                                    8 - indicated error or end-of-run
c                                   11 - error in user routine
c                                   12 - macro stop command
c                                   13 - job already running
c                                   14 - untrapped xox error
c                                   15 - anserr fatal error
c                                   16 - possible full disk
c                                   17 - possible corrupted or missing file
c                                   18 - Error in VM routines (corrupt db?)
c                                   21 - unauthorized code section entered
c                                   25 - unable to open x11 server
c                                   30 - quit signal
c                                   31 - failure to get signal in max time
c                                           (syhold)
c                                  >32 - system dependent error
c                                   35 - fatal error on another process
c                                        (distributed ANSYS)
c
c  output arguments:  none
```

4.3. Vector Functions

4.3.1. Function vdot (Computing the Dot Product of Two Vectors)

```
*deck,vdot
      function  vdot (v1,v2,n)
c *** primary function: compute dot product of vectors v1 and v2

c *** Notice - This file contains ANSYS Confidential information ***

c  input arguments:
c     v1       (dp,ar(n),in)     - vector v1
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c     v2       (dp,ar(n),in)     - vector v2
c     n        (int,sc,in)       - length of vectors v1 and v2

c  output arguments:
c     vdot     (dp,sc,out)       - dot product of v1 and v2

c
```

4.3.2. Function vidot (Computing the Dot Product of Two Vectors with Increments)

```
*deck,vidot
      function vidot (v1,inc1,v2,inc2,n)
c *** primary function: compute the dot product of vectors v1 and v2
c *** Notice - This file contains ANSYS Confidential information ***
c
c     ---- inc1 and inc2 must be positive!
c
c  input arguments:
c     v1      (dp,ar,in)           - vector 1
c     inc1    (int,sc,in)          - number of rows in vector 1
c     v2      (dp,ar,in)           - vector 2
c     inc2    (int,sc,in)          - number of rows in vector 2
c     n       (int,sc,in)          - length of vectors
c  output arguments:
c     vidot   (dp,func,out)    - dot product of vectors v1 and v2
```

4.3.3. Function vsum (Summing Vector Components)

```
*deck,vsum
      function vsum (va,n)
c *** primary function: sum the components of a vector

c *** Notice - This file contains ANSYS Confidential information ***

c  input arguments:
c     va       (dp,ar(n),in)     - vector va
c     n        (int,sc,in)       - length of vector va

c  output arguments:
c     vsum     (dp,sc,out)       - vector sum
```

4.3.4. Function vmax (Retrieving the Maximum Vector Value at a Given Location)

```
*deck,vmax
      function vmax (v,n,locmax)
c *** primary function: get the biggest value and location in v
c *** Notice - This file contains ANSYS Confidential information ***
c  input arguments:
c     v          (dp,ar,in)           - input vector v
c     n          (int,sc,in)          - length of input vector
c  output arguments:
c     vmax       (dp,fanc,out)        - biggest value in vector v
c     locmax     (int,sc,out)         - location of biggest value
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.3.5. Function lastv (Retrieving the Position of the Last Nonzero Term in a Double Precision Vector)

```
*deck,lastv
      function lastv (v,n)
c ********* find position of last non-zero term in a d.p. vector *********
c *** Notice - This file contains ANSYS Confidential information ***
c
c  input arguments:
c     v     (dp,ar(*),in)           - input vector
c     n      (int,sc,in)           - length of vector
c  output arguments:
c     lastv     (int,func,out)       - position of last non-zero term
```

4.3.6. Function izero (Setting an Integer Vector to Zero)

```
*deck,IZERO
      subroutine IZERO (ivect,n)
c **********  set an integer vector to zero  **********
```

4.3.7. Function imove (Assigning Equal Values to Two Integer Vectors)

```
*deck,imove
      subroutine imove (i1,i2,n)
c **********  move a vector from one to another  **********
c *** Notice - This file contains ANSYS Confidential information ***
c  input arguments:
c     i1     (int,ar,in)       - input vector
c     n      (int,sc,in)           - size of vector
c  output arguments:
c     i2     (int,ar,out)       - output vector
```

4.3.8. Subroutine vzero (Initializing a Vector to Zero)

```
*deck,vzero
      subroutine vzero (v,n)
c *** primary function:    initialize a vector to zero
c *** Notice - This file contains ANSYS Confidential information ***

c  input arguments:
c     n   (int,sc,in)   - the size of vector
c  output arguments:
c     v   (dp,ar,out)   - the vector need to be initialized
```

4.3.9. Subroutine vmove (Moving One Vector into Another)

```
*deck,vmove
      subroutine vmove (v1,v2,n)
c *** primary function: copy v1 vector into vector 2
c *** Notice - This file contains ANSYS Confidential information ***
c  input arguments:
c     v1      (dp,ar,in)           - vector 1
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c     n       (int,sc,in)          - length of vectors
c  output arguments:
c     v2      (dp,ar,out)          - vector 2
```

4.3.10. Subroutine vimove (Moving One Vector into Another Incrementally)

```
*deck,vimove
      subroutine vimove (v1,inc1,v2,inc2,n)
c *** primary function: move vector 1 into vector 2
c *** Notice - This file contains ANSYS Confidential information ***
c  input arguments:
c     v1      (dp,ar,in)           - vector 1
c     inc1    (int,sc,in)          - number of rows in vector 1
c     inc2    (int,sc,in)          - number of rows in vector 2
c     n       (int,sc,in)          - length of vector
c  output arguments:
c     v2      (dp,ar,out)          - vector 2
```

4.3.11. Subroutine vinit (Assigning a Scalar Constant to a Vector)

```
*deck,vinit
      subroutine vinit (v,n,const)
c *** primary function: initialize a vector to a constant
c *** Notice - This file contains ANSYS Confidential information ***
c  input arguments:
c     n           (int,sc,in)           - length of vector
c     const       (dp,sc,in)            - constant
c  output arguments:
c     v           (dp,ar,out)           - initialized vector
```

4.3.12. Subroutine viinit (Assigning a Scalar Constant to a Vector Incrementally)

```
*deck,viinit
      subroutine viinit (v,inc,n,const)
c *** primary function: set the components of vector v to const by increments
c *** Notice - This file contains ANSYS Confidential information ***
c  input arguments:
c     inc         (int,sc,in)           - number of rows in input vector
c     n           (int,sc,in)           - length of input vector
c     const       (dp,sc,in)            - const
c  output arguments:
c     v           (dp,ar,out)           - result vector
```

4.3.13. Subroutine vapb (Setting a Vector to Sum of Two vectors)

```
*deck,vapb
      subroutine vapb (a,b,c,n)
c *** primary function: add vector a to vector b to get vector c
c *** Notice - This file contains ANSYS Confidential information ***
c     input arguments:
c        a     (dp,ar,in)      - vector a
c        b     (dp,ar,in)      - vector b
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c        n     (int,sc,in)     - length of vectors
c     output arguments:
c        c     (dp,ar,out)     - vector c
```

4.3.14. Subroutine vapb1 (Combining Two Vectors in One)

```
*deck,vapb1
      subroutine vapb1 (a,b,n)
c *** primary function: add vector b to vector a and store in vector a
c  input arguments:
c     b       (dp,ar,in)           - vector b
c     n       (int,sc,in)          - length of vectors
c  output arguments:
c     a       (dp,ar,inout)        - vector a
```

4.3.15. Subroutine vapcb1 (Multiplying a Vector to a Constant)

```
*deck,vapcb1
      subroutine vapcb1 (a,b,n,const)
c *** primary function: multiply vector b to constant, add to vector a,
c                       and store in vector a
c  input arguments:
c     b       (dp,ar,in)           - vector b
c     n       (int,sc,in)          - length of vectors
c     const   (dp,sc,in)           - constant
c  output arguments:
c     a       (dp,ar,inout)        - vector a
```

4.3.16. Subroutine vamb (Gets a Third Vector by Subtracting One Vector from Another)

```
*deck,vamb
      subroutine vamb (a,b,c,n)
c *** primary function: subtract vector b from vector a to get vector c
c *** Notice - This file contains ANSYS Confidential information ***
c  input arguments:
c     a       (dp,ar,in)           -  vector a
c     b       (dp,ar,in)           -  vector b
c     n       (int,sc,in)           - size of vectors
c  output arguments:
c     c       (dp,ar,out)       - vector c
```

4.3.17. Subroutine vamb1 (Subtracting One Vector from Another)

```
*deck,vamb1
      subroutine vamb1 (a,b,n)
c *** primary function: subtract vector b from vector a and save in vector a
c *** Notice - This file contains ANSYS Confidential information ***
c  input arguments:
c     b       (dp,ar,in)           - vector b
c     n       (int,sc,in)           - size of vectors
c  output arguments:
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c     a       (dp,ar,inout)        - vector a
```

4.3.18. Subroutine vmult (Multiplying a Vector by a Constant)

```
*deck,vmult
      subroutine vmult (v1,v2,n,const)
c *** primary function: multiply a vector by a constant
c *** Notice - This file contains ANSYS Confidential information ***
c  input arguments:
c     v1       (dp,ar,in)          - input vector
c     n        (int,sc,in)         - length of vectors
c     const    (dp,sc,in)          - constant
c  output arguments:
c     v2       (dp,ar,out)         - result vector
```

4.3.19. Subroutine vmult1 (Multiplying a Vector by a Constant)

```
*deck,vmult1
      subroutine vmult1 (v1,n,const)
c *** primary function:  multiply a vector by a constant
c *** Notice - This file contains ANSYS Confidential information ***
c  input arguments:
c     n       (int,sc,in)   - size of vector
c     const   (dp,sc,in)    - constant
c  output arguments:
c     v1      (dp,ar,out)   - result vector
```

4.3.20. Subroutine vcross (Defining a Vector via a Cross Product)

```
*deck,vcross
      subroutine vcross (a,b,c)

c primary function: calculate c = a x b

c *** Notice - This file contains ANSYS Confidential information ***

c     typ=int,dp,log,chr,dcp   siz=sc,ar(n)   intent=in,out,inout

c  input arguments:
c     a        (dp,ar(3),in)     - first vector to be cross-multiplied
c     b        (dp,ar(3),in)     - second vector to be cross-multiplied

c  output arguments:
c     c        (dp,ar(3),out)    - resulting vector
c
c
```

4.3.21. Subroutine vnorme (Normalizing a Three-Component Vector)

```
*deck,vnorme
      subroutine vnorme (iel,v)
c primary function:  normalize a vector to unit length
c  this routine is to be called only from the elements.   it is only
c  for a three component vector(i.e. processing geometry).
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c  this routine also differs from vnorm in that an error message is called
c  if the vector length is zero.


c *** Notice - This file contains ANSYS Confidential information ***

c  input arguments:
c     iel      (int,sc,inout)    - element number
c     v        (dp,ar(3),inout)  - vector to be normalized

c  output arguments:
c     iel      (int,sc,inout)    - if 0, vector has zero length
c     v        (dp,ar(3),inout)  - normalized vector
```

4.3.22. Subroutine vnorm (Normalizing a Vector to Unit Length)

```
*deck,vnorm
      subroutine vnorm (v,n)
c *** primary function: normalize a vector to unit length

c *** Notice - This file contains ANSYS Confidential information ***

c  input arguments:
c     v        (dp,ar(n),inout)  - vector v
c     n        (int,sc,inout)    - dimension length of vector v

c  output arguments:
c     v        (dp,ar(n),inout)  - normalized vector v
c     n        (int,sc,inout)    - n = 0 if error in operation
```

4.3.23. Function ndgxyz (Getting the X,Y,Z Vector for a Node)

```
*deck,ndgxyz
      function ndgxyz (node,xyz)
c *** primary function:    get x,y,z vector for a node.

c *** Notice - This file contains ANSYS Confidential information ***

c     input arguments:
c        node     (int,sc,in)       - node number for operation.

c     output arguments:
c        ndgxyz   (int,sc,out)      - status of node.
c                                        0=node is undefined.
c                                       -1=node is unselected.
c                                        1=node is selected.
c        xyz      (dp,ar(3),out)    - vector containing x,y,z
```

4.3.24. Subroutine ndpxyz (Storing X,Y,Z for a Node)

```
*deck,ndpxyz
      subroutine ndpxyz (node,xyz)
c *** primary function:    store x,y,z vector for a node.

c *** Notice - This file contains ANSYS Confidential information ***
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c     input arguments:
c        node     (int,sc,in)       - node number for operation.
c        xyz      (dp,ar(3),in)     - vector containing x,y,z
c                                      (vector should be in global system)

c     output arguments:  none
```

4.4. Matrix Subroutines

4.4.1. Subroutine maxv (Multiplying a Vector by a Matrix)

```
*deck,maxv
      subroutine maxv (a,v,w, nr,nc)
c *** primary function: multiply a matrix by a vector

c *** Notice - This file contains ANSYS Confidential information ***

c  input arguments:
c     a        (dp,ar(nr,*),in)  - matrix a
c     v        (dp,ar(*),in)     - vector v
c     nr       (int,sc,in)       - number of rows in matrix a
c     nc       (int,sc,in)       - number of columns to multiply in matrix a

c  output arguments:
c     w        (dp,ar(*),out)    - product vector w
c
c *** mpg w = A v : A(nr,nc) : matrix vector product
c
```

4.4.2. Subroutine maxv1 (Multiplying a Vector by a Matrix)

```
*deck,maxv1
      subroutine maxv1 (a,v, nr,nc)
c *** primary function: multiply a vector by a matrix

c *** Notice - This file contains ANSYS Confidential information ***

c  input arguments:
c     a        (dp,ar(nr,nc),in) - matrix a
c     v        (dp,ar(nc),inout) - vector v
c     nr       (int,sc,in)       - number of rows in matrix a
c                                  *** nr limited to 60 ***
c     nc       (int,sc,in)       - number of columns to multiply in matrix a

c  output arguments:
c     v        (dp,ar(nr),inout) - product, stored in vector v
c
c *** mpg v = A v : A(nr,nc) : matrix vector product, max 60 rows
c
```

4.4.3. Subroutine matxv (Multiplying a Vector by a Full Transposed Matrix)

```
*deck,matxv
      subroutine matxv (a,v,w, nr,nc)
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c *** primary function: multiply vector by full transposed matrix

c *** Notice - This file contains ANSYS Confidential information ***

c  input arguments:
c     a        (dp,ar(nr,*),in)   - matrix a (first dimension must = nr)
c     v        (dp,ar(nv),in)     - vector v (nv must be greater or equal
c                                                    to nr)
c     nr       (int,sc,in)        - first dimension and number of active
c                                      rows of the untransposed matrix a
c                                      (also the number of active rows
c                                       of vector v)
c     nc       (int,sc,in)        - number of columns of the untransposed
c                                      matrix a
c                                      (also the number of computed items
c                                      in the product vector w)
c                                      if negative, accumulate

c  output arguments:
c     w        (dp,ar(na,*),out)  - product vector w
c
c
c *** mpg A(nr,nc) : matrix transpose vector product
c         w =     A+ v : if nr > 0
c         w = w + A+ v : if nr < 0
c
```

4.4.4. Subroutine matxv1 (Multiplying a Vector by a Full Transposed Matrix)

```
*deck,matxv1
      subroutine matxv1 (a,v, nr,nc)
c *** primary function: multiply vector by full transposed matrix

c *** Notice - This file contains ANSYS Confidential information ***

c  input arguments:
c     a        (dp,ar(nr,*),in)   - matrix a
c     v        (dp,ar(nr),inout)  - vector v
c     nr       (int,sc,in)        - number of rows in matrix (un-transposed)
c     nc       (int,sc,in)        - number of columns in matrix (un-transposed)
c                                     *** nc limited to 60 ***

c  output arguments:
c     v        (dp,ar(nc),inout)  - product, stored in vector v

c
c *** mpg A(nr,nc) : matrix transpose vector product
c         v = A+ v : max 60 nc
```

4.4.5. Subroutine matxb (Transposing a matrix)

```
*deck,matxb
      subroutine matxb (a,b,c, na,nb,nc, n1,n2,n3)
c *** primary function:  (a)t * (b) = (c)     t means transpose

c *** Notice - This file contains ANSYS Confidential information ***

c  input arguments:
c     a        (dp,ar(na,*),in)   - matrix a
c     b        (dp,ar(nb,*),in)   - matrix b
c     na       (int,sc,in)        - number of rows in matrix a
c     nb       (int,sc,in)        - number of rows in matrix b
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c     nc       (int,sc,in)        - number of rows in matrix c
c     n1       (int,sc,in)        - number of rows in matrix c to fill
c     n2       (int,sc,in)        - number of columns in matrix c to fill
c     n3       (int,sc,in)        - number of rows in matrix a and
c                                   number of rows of matrix b
c                                   to work with (the two need
c                                   to be the same for the inner product)
c                                   if n3 is negative, accumulate results in c

c  output arguments:
c     c        (dp,ar(nc,*),out)  - product matrix c

c *** mpg C =     A+ B     if n3 > 0
c         C = C + A+ B     if n3 < 0
c         A(na,*) B(nb,*) C(nc,*)  C:minor n1 * n2  n3: dot length
c
```

4.4.6. Subroutine maat (Changing a Matrix Value via Addition, Multiplication, and Transposition)

```
*deck,maat
      subroutine maat(a,c, nc,n, con)
c primary function: does con*a*at and sums the result onto c (a is a vector)

c *** Notice - This file contains ANSYS Confidential information ***

c     typ=int,dp,log,chr,dcp   siz=sc,ar(n)   intent=in,out,inout

c  input arguments:
c     a        (dp,ar(*),in)        - vector to be multiplied by itself to
c                                     generate an nxn square matrix
c                                      (a by a-transposed)
c     c        (dp,ar(nc,*),inout) - matrix to be accumulated onto
c     nc       (int,sc,in)         - number of rows in the c matrix
c     n        (int,sc,in)         - size of square matrix
c     con      (dp,sc,in)          - multiplier on above square matrix

c  output arguments:
c     c        (dp,ar(nc,*),inout) - matrix to be accumulated onto
c                                     only the lower triangular matrix is done

c         Note:  this routine is usually followed by matsym,
c                                              to do the complete matrix
c
```

4.4.7. Subroutine matba (Updating Matrix Value via Transposition, Multiplications, and Addition)

```
*deck,matba
      subroutine matba (a,b,c,na,nb,nc,n1,n2,work,con)
c primary function:     does con(at*b*a) and sums the result
c
c *** Notice - This file contains ANSYS Confidential information ***
c  input arguments:
c     a        (dp,ar(na,*),in)   - matrix a
c     b        (dp,ar(nb,*),in)   - matrix b (must be square,
c                                          and maximum dimension is (15,15)
c     c        (dp,ar(nc,*),inout)- matrix c (see output)
c     na       (int,sc,in)        - number of rows in matrix a
c     nb       (int,sc,in)        - number of rows in matrix b
c     nc       (int,sc,in)        - number of rows in matrix c
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c     n1       (int,sc,in)        - number of rows in matrix a and
c                                   number of rows of matrix b
c                                   to work with (the two need
c                                   to be the same for the inner product)
c     n2       (int,sc,in)        - number of columns in matrix c to fill
c     con      (dp,sc,in)         - multiplier on product added to sum

c  output arguments:
c     c        (dp,ar(nc,*),inout)- c = c + con*at*b*a
c     work     (dp,ar(n2,*),out)  - at*b (this byproduct is occasionally useful)

c *** mpg C = C + con A+ B A  A(na,*) B(nb,*) C(nc,*)  C:minor n1 * n2
c     see matbabd for block diagonal
c
```

4.4.8. Subroutine matsym (Filling the Upper Triangle from the Lower Triangle)

```
*deck,matsym
      subroutine matsym (a,nd,n)
c primary function:   fill upper triangle from lower triangle

c *** Notice - This file contains ANSYS Confidential information ***

c     typ=int,dp,log,chr,dcp   siz=sc,ar(n)   intent=in,out,inout

c  input arguments:
c     a        (dp,ar(nd,*),inout) - matrix to have its lower triangular part
c                                     copied to its upper triangular part
c     nd       (int,sc,in)         - number of rows of the a matrix
c     n        (int,sc,in)         - size of matrix to be processed

c  output arguments:
c     a        (dp,ar(nd,*),inout) - matrix that has its lower triangular part
c                                     copied to its upper triangular part
c
```

4.4.9. Subroutine mctac (Transposing a symmetric matrix)

```
*deck,mctac
      subroutine mctac (a,na,c,nc,nold,nnew)
c  **** function: do  a = c(transpose) * a * c ,  where a is symmetric  **

c *** Notice - This file contains ANSYS Confidential information ***


c  input arguments:
c     a        (dp,ar(na,na),inout) matrix to be pre and post multiplied
c                                     (part operated on must be
c                                     square(nold x nold) and symmetric)
c     na       (int,sc,in)          first dimension of the a matrix
c     c        (dp,ar(nc,nnew),in)  matrix to pre and post multiply a by
c                                     (part used may be rectangular(nold x nnew))
c     nc       (int,sc,in)          first dimension of the c matrix
c     nold     (int,sc,in)          size of part of 'A' matrix that is
c                                     to be processed(input size).   maximum = 64
c     nnew     (int,sc,in)          size of part of 'A' matrix that
c                                     results from this operation(output size).
c                                                            maximum = 64

c  output arguments:
c     a        (dp,ar(na,na),inout) resulting matrix
c                                     (still square(nnew x nnew) and symmetric).
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.4.10. Subroutine tran (Transposing a matrix)

```
*deck,tran
      subroutine tran (zs,tr,nz,ntr,nrow,irot)
c primary function: perform   tr-transpose * zs * tr ************

c *** Notice - This file contains ANSYS Confidential information ***

c  input arguments:
c     variable (typ,siz,intent)    description
c     zs       (dp,ar(nz,nz),inout) - matrix to be transformed
c     tr       (dp,ar(ntr,ntr),in)  - transformation matrix
c     nz       (int,sc,in)          - dimensioned size of zs matrix
c     ntr      (int,sc,in)          - dimensioned size of tr matrix
c     nrow     (int,sc,in)          - number of rows of zs matrix to transform
c     irot     (int,sc,in)          - block size to transform(size of tr matrix)

c  output arguments:
c     variable (typ,siz,intent)    description
c     zs       (dp,ar(nz,nz),inout) - transformed matrix
```

4.4.11. Subroutine symeqn (Solving Simultaneous Linear Equations)

```
*deck,symeqn
      function symeqn (a,nd,n,nc,defFlag)
c
c primary function:  solve a set of simultaneous linear equations
c
c secondary functions: invert a matrix
c
c          NOTE:  this routine assumes that the matrix to be solved or
c                 inverted is positive or negative definite.  This routine
c                 also assumes that the diagonals are all non-zero. If
c                 this assumption is not true, use isimeq.F.
c
c *** Notice - This file contains ANSYS Confidential information ***
c
c  input arguments:
c     variable (typ,siz,intent)    description
c     a        (dp,ar(nd,*),inout) - matrix to be solved or inverted
c                                       second dimension must be at least:
c                                                          n + abs(nc)
c     nd       (int,sc,in)         - first dimension of the a matrix
c     n        (int,sc,in)         - number of equations
c     nc       (int,sc,in)         - number of additional columns.
c                                       if nc = +n or -n, invert n x n matrix and
c                                       put result in the n+1 to 2xn columns.
c                                       if nc is 0 or negative, nc will be reset to
c                                       n and then symeqn will set up identity
c                                       matrix after the input matrix, where the
c                                       result of the inversion will be put.
c                                       if nc is positive and less than n, do a
c                                       partial inversion. see example 1 below.
c     defFlag  (int,sc,in)         - flag indicating that incoming matrix MUST be:
c                                       -1 - negative definite
c                                        0 - positive or negative definite
c                                        1 - positive definite
c
c  output arguments:
c     variable (typ,siz,intent)    description
```

```
c     symeqn   (in,sc,out)         - 0 - non-singular matrix
c                                    1 - singular matrix
c                                    2 - near-singular matrix
c     a        (dp,ar(nd,*),inout) - results or inverted matrix.
c                                      starts in column n+1.
c                                      note:  original information is destroyed.
c
c   example 1:   Solve three simultaneous linear equations:
c                      i = symeqn (a(1,1),3,3,1)
c                   calling routine has a dimensioned as a(3,4)
c                   each equation has its 3 coefficents in the first 3 columns,
c                    and the constant term is in the fourth column.
c                   solution is in fourth column.
c
c   example 2:   Invert a 3x3 matrix:
c                      i = symeqn (a(1,1),3,3,-3)
c                   calling routine has a dimensioned as a(3,6)
c                   input matrix was input in first 3 columns
c                   output matrix in ouput in last 3 columns
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 5: Using Python to Code UPF Subroutines

As an alternative to compiled languages like C and Fortran, you can use the Python language to code user programmable subroutines. A subset of the documented UPF subroutines support the Python UPF capability (see Supported UPF Subroutines (p. 371)).

You must install a Python distribution before using this feature. Python 3.6 through Python 3.7 are supported.

Python UPFs are only supported on Linux.

It is strongly recommended you start your code based on one of the examples in Python UPF Examples (p. 375). In your Python code, you can make use of standard Python libraries like NumPy.

The following topics are available:

5.1. Supported UPF Subroutines

A subset of the entire set of available UPF subroutines support Python coding (see the table below). This list will be expanded in the future.

For more information about each subroutine, click the link to the Fortran description.

Table 5.1: Python Support for Subroutines

Subroutine	Fortran Description
Material Behavior	
`UserMat`	Subroutine UserMat (Creating Your Own Material Model) (p. 205)
`UserMatTh`	Subroutine UserMatTh (Creating Your Own Thermal Material Model) (p. 217)
`UserHyper`	Subroutine UserHyper (Writing Your Own Isotropic Hyperelasticity Laws) (p. 223)
`UserCreep`	Subroutine UserCreep (Defining Creep Material Behavior) (p. 227)
Modifying and Monitoring Elements	
`UsrShift`	Subroutine UsrShift (Calculating Pseudotime Time Increment) (p. 203)

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Subroutine	Fortran Description
UTimeInc	Subroutine UTimeInc (Overriding the Program-Determined Time Step) (p. 203)
UCnvrg	Subroutine UCnvrg (Overriding the Program-Determined Convergence) (p. 204)
Customizing Loads	
usrefl	Subroutine usrefl (Changing Scalar Fields to User-Defined Values) (p. 252)
userpr	Subroutine userpr (Changing Element Pressure Information) (p. 253)
usercv	Subroutine usercv (Changing Element Face Convection Surface Information) (p. 254)
userfx	Subroutine userfx (Changing Element Face Heat Flux Surface Information) (p. 255)
Access Subroutines	
UanBeg / UanFin	Access at the Beginning and End of Various Operations (p. 277)
USolBeg / USolFin	
ULdBeg / ULdFin	
UItBeg / UItFin	
USsBeg / USsFin	

5.2. Python UPF Methodology

Coding a Python UPF is different from using a compiled language like C/C++ or Fortran, mainly in terms of the API. Because the gRPC technology is used to handle the communication and the exchange of data between the Python process and the Mechanical APDL process, you need to understand the way this feature handles the serialization/deserialization of data.

The main difference is in the subroutine arguments. Instead of having a full list of arguments as described for each of the subroutines, there are only two: the request object (for inputs), and the response object (for outputs). If an argument is both input and output of the subroutine, it will be part of both objects.

The description of the request object and the response object can be found in the `MapdlUser.proto` file stored in this installation directory:

```
Ansys Inc\v212\ansys\syslib\ansGRPC\User
```

First, create a Python file starting from this template:

my_upf.py

```
import grpc
import sys
from mapdl import *

class MapdlUserService( MapdlUser_pb2_grpc.MapdlUserServiceServicer ):

#   ######################################################################
    def UAnBeg( self, request, context):
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
        print( " ======================================= ")
        print( " >> Inside the PYTHON UAnBeg routine  << ")
        print( " ======================================= \n")

        response = google_dot_protobuf_dot_empty__pb2._EMPTY()
        return response

if __name__ == '__main__':
    upf.launch( sys.argv[0])
```

Note that the Mechanical APDL application automatically installs a Mechanical APDL Python package (a set of Python functions to handle the connection between Mechanical APDL and the Python environment). Each Python UPF must be imported, as noted by:

```
from mapdl import *
```

The above example redefines the UAnBeg routine and prints a customized banner. This file must be in the same directory as the input file.

To use this Python UPF, you must add the Mechanical APDL /UPF command (p. 134) to your input file:

my_inp.dat

```
 /UPF,my_upf.py

 ! The UAnBeg UPF must be activated by using the USRCAL APDL command

 USRCAL,UANBEG
```

This command is trapped by the Mechanical APDL Launcher so that a Python gRPC server is up and running when the Mechanical APDL process starts.

When launching Mechanical APDL using this input file, you will see the following printout to indicate Mechanical APDL detected the Python UPF instructions and has launched a Python server:

```
Processing "/upf" found in input file "my_inp.dat"

 Python UPF Detected

PYTHON VERSION : 3.6
>>
>> START PYTHON GRPC SERVER
>>
>> User Functions Python File :  my_upf.py
>>
>> Server started on port [50054]
```

During the Mechanical APDL process, you will see this Python printout:

```
RUN SETUP PROCEDURE FROM FILE= /ansys_inc/v212/ansys/apdl/start.ans
 =======================================
 >> Inside the PYTHON UAnBeg routine  <<
 =======================================
```

At the very end of the process, the Python server is automatically shutdown:

```
…
|---------------------------------------------------------------|
|                                                               |
|   CP Time      (sec) =          0.326       Time  =  10:40:24   |
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
|   Elapsed Time (sec) =          2.000       Date  =  03/11/2021 |
|                                                                  |
*------------------------------------------------------------------*

>> We shutdown Python Server(s)
```

5.3. Accessing the Database from the Python Code

Within your UPF routine, you may need to access the Mechanical APDL database in read/write mode.

In the Python code, you can create a connection with the DB server. This command must be called only once, so you can protect the call based on the value of a static variable:

```
import grpc
import sys
from mapdl import *

firstcall = 1

class MapdlUserService(MapdlUser_pb2_grpc.MapdlUserServiceServicer):


#   ###################################################################
    def UserMat( self, request, context):

        global firstcall

        if firstcall == 1:
            print( ">> Connection to the MAPDL DB Server\n")
            db.start()
            firstcall = 0

        # continuation of the python function
        # …
```

Once the DB connection has been initialized, you can access the database of the Mechanical APDL instance in read/write mode.

A subset of the functions documented in Accessing the Mechanical APDL Database (p. 289) have been exposed and can be called from the Python code. Below is a list of those functions:

Supported Database Access Functions	
`db.start()`	Initializes the connection with a running Mechanical APDL instance. The DB Server is automatically started in Mechanical APDL if a **/UPF** command with a python file has been detected.
`db.stop()`	Closes the connection with the DB Server.
`db.ndnext(next)`	Equivalent to the function described in Function ndnext (Getting the Next Node Number) (p. 290)
`db.ndinqr(ind, key)`	Equivalent to the function described in Function ndinqr (Getting Information About a Node) (p. 293)
`db.getnod(inod)`	Equivalent to the function described in Function getnod (Getting a Nodal Point) (p. 294)
`db.putnod(inod, x, y, z)`	Equivalent to the function described in Function putnod (Storing a Node) (p. 294)

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

`db.elnext(ielm)`	Equivalent to the function described in Function elnext (Getting the Number of the Next Element) (p. 291)
`db.getelem(ielm)`	Equivalent to the function described in Function elmget (Getting an Element's Attributes and Nodes) (p. 297)
`db.get_ElmInfo(inquire)`	Equivalent to the function get_ElmInfo described in Accessing Solution and Material Data (p. 403)
`db.get_ElmData(kchar, elemId, kMatRecPt, ncomp, vect)`	Equivalent to the function get_ElmData described in Accessing Solution and Material Data (p. 403)
`db.putElmData(inquire, elemId, kIntg, nvect, vect)`	Equivalent to the function put_ElmData described in Accessing Solution and Material Data (p. 403)

5.4. Python UPF Limitations

The Python UPF capability has these limitations:

- Currently, Distributed Ansys is not supported. You must specify the -smp option on the command line to make sure Mechanical APDL is running in shared-memory processing mode.
- Python UPFs are only available on Linux platforms.

5.5. Python UPF Examples

The following Python UPF Examples are available:

5.5.1. Example: Python `UserMat` Subroutine

This example simulates a block modeled with 3-D elements. The `usermat.py` user material is equivalent to linear elastic. The block is under uniaxial compression. The final deformation is compared with the theoretical result.

5.5.1.1. Input Data

```
/batch,list
/title,upf-py1s, test usermat.py with 3-D elements

/prep7
/upf,usermat.py
tb,user,1,,2
tbdata,1,1e5, 0.3    ! E, Poisson

et,1,185

block,0,1,0,1,0,1
esize,1
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
mshape,0,3D
vmesh,all
elist

nsel,s,loc,x,0
d,all,ux

nsel,s,loc,y,0
d,all,uy

nsel,s,loc,z,0
d,all,uz,

allsel,all
finish

/solu

time,1
deltime,0.1
eresx,no
nsel,s,loc,x,1
!d,all,ux,-0.01
sf,all,pres,1000        ! pressure on x-axis
allsel,all

outres,all,all

solve

finish
/POST1
set,last
esel,s,elem,,1
/output
presol,s
presol,epel
/com, expected results: Sx=-1000, epel_x=-1e-2
finish
/exit,nosave
```

5.5.1.2. usermat.py

```
import grpc
import sys
import math
import numpy as np
from mapdl import *

class MapdlUserService( MapdlUser_pb2_grpc.MapdlUserServiceServicer ):


#   ###################################################################
    def UserMat(self, request, context):

        ncomp = request.ncomp
        nDirect = request.nDirect

        response = MapdlUser_pb2.UserMatResponse()

        response.stress[:] = request.stress[:]
        response.ustatev[:] = request.ustatev[:]
        response.sedEl = request.sedEl
        response.sedPl = request.sedPl
        response.epseq = request.epseq
        response.epsPl[:] = request.epsPl[:]
        response.var0 = request.var0
        response.var3 = request.var3
        response.var4 = request.var4
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
        response.var5 = request.var5
        response.var6 = request.var6
        response.var7 = request.var7

        if ncomp > 4:                        # ***    3d, plane strain and axisymmetric example
            usermat3d( request, context, response)
        elif nDirect== 2 and ncomp == 3:     # ***    plane stress example
            usermatps( request, context, response)
        elif ncomp == 3:                     # ***    3d beam example
            usermatbm( request, context, response)
        elif ncomp == 1:                     # ***    1d beam example
            usermat1d( request, context, response)

        return response



def usermat3d( request, context, response):

    ZERO       = 0.
    HALF       = 0.5
    THIRD      = 1./3.
    ONE        = 1.
    TWO        = 2.
    SMALL      = 1.e-08
    sqTiny     = 1.e-20
    ONEDM02    = 1.e-02
    ONEDM05    = 1.e-05
    ONEHALF    = 1.5
    TWOTHIRD   = 2.0/3.0
    mcomp      = 6

    G = [1., 1., 1., 0., 0. ,0.]

    db.start()                          # Connect to the MAPDL DB gRPC Server
    ncomp = request.ncomp

    # *** get Young's modulus and Poisson's ratio
    young    = request.prop[0]
    posn     = request.prop[1]
    twoG     = young / (ONE+posn)
    elast1   = young*posn/((1.0+posn)*(1.0-TWO*posn))
    elast2   = HALF*twoG

    #
    # *** calculate elastic stiffness matrix (3d)
    #
    dsdeEl = np.zeros( ( 6, 6))

    dsdeEl[0,0] = (elast1+TWO*elast2)*G[0]*G[0]
    dsdeEl[0,1] = elast1*G[0]*G[1]+elast2*TWO*G[3]*G[3]
    dsdeEl[0,2] = elast1*G[0]*G[2]+elast2*TWO*G[4]*G[4]
    dsdeEl[0,3] = elast1*G[0]*G[3]+elast2*TWO*G[0]*G[3]
    dsdeEl[0,4] = elast1*G[0]*G[4]+elast2*TWO*G[0]*G[4]
    dsdeEl[0,5] = elast1*G[0]*G[5]+elast2*TWO*G[3]*G[4]

    dsdeEl[1,1] = (elast1+TWO*elast2)*G[1]*G[1]
    dsdeEl[1,2] = elast1*G[1]*G[2]+elast2*TWO*G[5]*G[5]
    dsdeEl[1,3] = elast1*G[1]*G[3]+elast2*TWO*G[0]*G[3]
    dsdeEl[1,4] = elast1*G[1]*G[4]+elast2*TWO*G[0]*G[4]
    dsdeEl[1,5] = elast1*G[1]*G[5]+elast2*TWO*G[1]*G[5]

    dsdeEl[2,2] = (elast1+TWO*elast2)*G[2]*G[2]
    dsdeEl[2,3] = elast1*G[2]*G[3]+elast2*TWO*G[4]*G[5]
    dsdeEl[2,4] = elast1*G[2]*G[4]+elast2*TWO*G[4]*G[2]
    dsdeEl[2,5] = elast1*G[2]*G[5]+elast2*TWO*G[5]*G[2]

    dsdeEl[3,3] = elast1*G[3]*G[3]+elast2*(G[0]*G[1]+G[3]*G[3])
    dsdeEl[3,4] = elast1*G[3]*G[4]+elast2*(G[0]*G[5]+G[4]*G[3])
    dsdeEl[3,5] = elast1*G[3]*G[5]+elast2*(G[3]*G[5]+G[4]*G[1])
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
    dsdeEl[4,4] = elast1*G[4]*G[4]+elast2*(G[0]*G[2]+G[4]*G[4])
    dsdeEl[4,5] = elast1*G[4]*G[5]+elast2*(G[3]*G[2]+G[4]*G[5])

    dsdeEl[5,5] = elast1*G[5]*G[5]+elast2*(G[1]*G[2]+G[5]*G[5])

    for i in range( 0, 5):
        for j in range( i+1, 6):
            dsdeEl[j,i] = dsdeEl[i,j]

    Strain = np.zeros( ncomp)
    Strain[0:ncomp] = request.Strain[0:ncomp]
    dStrain = np.zeros( ncomp)
    dStrain[0:ncomp] = request.dStrain[0:ncomp]

    #
    # *** calculate the stress and
    #     copy elastic moduli dsdeEl to material Jacobian matrix

    strainEl = np.copy(Strain)                  # strainEl = Strain
    strainEl = np.add( strainEl, dStrain)       # strainEl += dStrain

    dsdePl = np.copy(dsdeEl)
    sigElp = np.zeros ( ncomp)
    sigElp = dsdeEl.dot( strainEl)

    response.stress[:] = sigElp
    dsdePl.shape = (6*6)
    response.dsdePl[:] = dsdePl

    return response

if __name__ == '__main__':
    upf.launch( sys.argv[0])
```

5.5.2. Example: Python `UsrShift` Subroutine

This example describes a block of Prony viscoplastic material with a user-defined shift function following a Tool-Narayanaswamy shift function. Uniaxial tension is applied on one end and held for 15 seconds with a constant 280 K uniform temperature. The final stress is obtained to check stress relaxation.

5.5.2.1. Input Data

```
/batch,list
/title,upf-py10s, test usrshift.py
/com
/com
/com
/nopr

/prep7
/upf,usrshift.py

n1=60
n2=n1*10
n3=n1
dy = 0.0045
fact=2
t1end=30.0/fact
alpha = 0.5
tau = 2.0
a1 = alpha          ! participating factor for el182, 183
t1 = tau
c1 = a1/a1          ! participating factor for el88

tr = 0
theta = 280
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
toffst,273
tunif, theta
tref,0
b1 = log(fact)*(273+tr)*(273+theta)/(theta-tr)
b2 = 1
b11=b1/273/273

young = 20e5
poiss = 0.3
G0 = young/2/(1+poiss)
K0 = young/3/(1-2*poiss)

! material 1                ! rate-dependent vpl
mp,ex,1,young
mp,nuxy,1,0.3
tb,prony,1,,1,shear         ! define viscousity parameters
tbdata,1,a1,t1
tb,prony,1,,1,bulk          ! define viscousity parameters
tbdata,1,a1,t1
tb,shift,1,,2,100           ! Tool-Narayanaswamy shift function
tbdata,1,tr,b11,

! FE model and mesh

et,1,186
mat,1
block,0,1,0,1,0,1
esize,1
vmesh,1

nall
nsel,s,loc,x
d,all,ux
nall
nsel,s,loc,y
d,all,uy
nall
nsel,s,loc,z
d,all,uz

/solu
nlgeom,on
cnvtol,u,,1.0e-8
cnvtol,f,,1.0e-6
nsel,s,loc,y,1.000
d,all,uy,dy
nall
time,1.0e-8
nsubst,1,1,1
outres,all,-10
solve

nsel,s,loc,y,1.000
time,t1end
d,all,uy,dy
nall
nsubst,n1,n2,n3
outres,all,-10
outpr,all,last
solve

finish

/post1
set,last
/output
presol,s

/com, expected results   Sy=4490.0
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
finish
/exit,nosave
```

5.5.2.2. usrshift.py

```
import grpc
import sys
import math
from mapdl import *

class MapdlUserService( MapdlUser_pb2_grpc.MapdlUserServiceServicer ):

#   ####################################################################

    def UsrShift(self, request, context):

        response = MapdlUser_pb2.UsrShiftResponse()
        one = 1.0
        half = 0.5
        quart = 0.25

        tref = request.propsh[0]
        temp = request.temp
        timinc = request.timinc
        dtemp = request.dtemp
        nTerms = request.nTerms

        thalf = temp - dtemp*half - tref
        t3quart = temp - dtemp*quart - tref

        c1 = 0.0
        c2 = 0.0

        for i in range(nTerms-1):
            c1 = c1 + request.propsh[i+1] * thalf ** (i+1)
            c2 = c2 + request.propsh[i+1] * t3quart ** (i+1)

        dxi = math.exp(c1) * timinc
        dxihalf = math.exp(c2) * timinc * half

        response.dxi = dxi
        response.dxihalf = dxihalf

        return response

if __name__ == '__main__':
    upf.launch( sys.argv[0])
```

5.5.3. Example: Python UserHyper Subroutine

This example models a block under simple uniaxial tension. The block is made of a user-defined hyper material that is identical to Arruda-Boyce Hyperelasticity. Large deformation effects are included. The final stress is printed out to compare against the reference.

5.5.3.1. Input Data

```
/BATCH,LIST
/title, upf-py16s, test UserHyper.py with MAPDL
/com    displacement-controlled uniaxial tension test for Boyce material model

/prep7

/upf,userhyper.py
tb,hyper,1,,,user
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
tbdata,1,2/100,0.2,2.8284

et,1,185

block,0,1,0,1,0,1
esize,1
vmesh,1

nsel,s,loc,x
d,all,ux
nsel,s,loc,y
d,all,uy
nsel,s,loc,z
d,all,uz
nall

nsel,s,loc,x,1.0
d,all,ux,0.3

nall

/solu

nlgeom,on
time,1
nsubst,5,20,5

/out,scratch
solve

/post1
/output

set,1,last
presol,s,x

/com, expected results from equivalent userhyper.F
/com,    NODE     SX           SY           SZ           SXY          SYZ
/com,       2  0.20118      0.32054E-003 0.32054E-003 0.13752E-015 0.67903E-017
/com,       4  0.20118      0.32054E-003 0.32054E-003 0.13776E-015 0.40293E-017
/com,       3  0.20118      0.32054E-003 0.32054E-003 0.50933E-015-0.10653E-014
/com,       1  0.20118      0.32054E-003 0.32054E-003 0.50909E-015-0.54682E-015
/com,       5  0.20118      0.32054E-003 0.32054E-003-0.15222E-015 0.58245E-015
/com,       6  0.20118      0.32054E-003 0.32054E-003-0.15313E-015 0.10856E-014
/com,       7  0.20118      0.32054E-003 0.32054E-003-0.55356E-015 0.17421E-016
/com,       8  0.20118      0.32054E-003 0.32054E-003-0.55265E-015 0.28848E-016

finish
/exit,nosave
```

5.5.3.2. userhyper.py

```
import grpc
import sys
from mapdl import *
import math
import numpy as np

firstcall = 1

class MapdlUserService( MapdlUser_pb2_grpc.MapdlUserServiceServicer ):

    #   ######################################################################
    def UserHyper(self, request, context):

        global firstcall
        if firstcall == 1:
            print( ">> Using Python UserHyper function\n")
            firstcall = 0
```

```
        prophy = np.copy(request.prophy)
        invar = np.copy(request.invar)

        response = MapdlUser_pb2.UserHyperResponse()

        ZERO  = 0.0
        ONE   = 1.0
        HALF  = 0.5
        TWO   = 2.0
        THREE = 3.0
        TOLER = 1.0e-12

        ci = (0.5,0.05,.104761904761905E-01,.271428571428571E-02,.770315398886827E-03)

        i1   = invar[0]
        jj   = invar[2]
        mu   = prophy[1]
        lm   = prophy[2]
        oD1  = prophy[0]
        i1i  = ONE
        im1  = ONE/i1
        t3i  = ONE
        potential = ZERO
        pInvDer = np.zeros(9)

        for i in range(5):
           ia    = i+1
           t3i   = t3i * THREE
           i1i   = i1i * i1
           i1i1  = i1i  * im1
           i1i2  = i1i1 * im1
           lm2 = ci[i] / (lm ** (TWO*(ia-ONE)))
           potential = potential + lm2 * (i1i - t3i)
           pInvDer[0] = pInvDer[0] + lm2 * ia * i1i1
           pInvDer[2] = pInvDer[2] + lm2 * ia * (ia-ONE) * i1i2

        potential = potential * mu
        pInvDer[0] = pInvDer[0] * mu
        pInvDer[2] = pInvDer[2] * mu

        j1 = ONE / jj
        pInvDer[7] = ZERO
        pInvDer[8] = ZERO
        if oD1 > TOLER:
           oD1  = ONE / oD1
           incomp = False
           potential = potential + oD1*((jj*jj - ONE)*HALF - math.log(jj))
           pInvDer[7] = oD1*(jj - j1)
           pInvDer[8] = oD1*(ONE + j1*j1)

        response.potential = potential
        response.incomp = incomp
        response.pInvDer[:] = pInvDer[:]

        return response

if __name__ == '__main__':
    upf.launch( sys.argv[0])
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Appendix A. Creating External Commands in Linux

External commands allow you to add your own customized extensions to Mechanical APDL without relinking the program. You can create custom routines in C that access any of the Mechanical APDL API functions, link them into shared libraries using the supplied utilities, and execute the routines via the "external command" feature within Mechanical APDL. In addition, the program provides special commands that list all available external commands and allow you to reset all currently referenced external commands.

External command capability is supported on all Linux platforms. Refer to your Ansys, Inc. Linux Installation Guide for currently supported compilers; the following instructions assume the presence of compatible compilers and linkers.

A.1. Tasks in Creating an External Command

To create a functional external command, you will need to complete the following general steps:

- Create compilable source code.
- Create a shared library. This is facilitated by the `gen_share` utility and your system's **make** capability.
- Create an external table file (`ans_ext.tbl`), listing the various shared libraries, functions, and the related command.
- Set an environment variable pointing to the directory that holds the external table file.

The following sections detail each of these tasks.

A.1.1. Creating Compatible Code

You can create your functions using any of the API functions described in `//ansys_inc/v212/ansys/customize/include/cAnsInterface.h`, `cAnsQuery.h`, and `cAnsPick.h`. The following code segment demonstrates, at a minimal level, how to create functions that can be used as an entry point into a custom coded shared library.

The most important point in the following example it that the C program interface is an integer function that has one argument (a char pointer).

```
#include "cAnsInterface.h"
#include "CAnsQuery.h"
/*
-------------------------  Function Description --------------------
extfnc
  int extfnc(uecmd)
  char *uecmd;

Purpose:
     Demonstrate C API entry function for an external command.
```

```
Parameters:
     Input
     ----------------------------
       uecmd
         The ANSYS external command string.

     Output
     ----------------------------

Return Value:
     The return value is ignored by the calling function;

------------------------ End Function  Description ------------------
*/
int extfnc(char* uecmd)
{
   /* Note: uecmd is the entire command given to invoke this function */
    char* cmdsend = {"/COM, COMMAND SENT FROM EXTERNAL COMMAND"};
    char* querystr = {"NODE,,NUM,MAX"};
    char strrtn[32];
    int i, itype;
    double dblrtn;

    /* Send a simple command to be executed */
    i = cAnsSendCommand(cmdsend);

    /* Perform a simple query */
    i = cAnsGetValue(querystr,&dblrtn,strrtn,&itype);

    /* Display the value retrieved */
    cAns_printf("Max Node Number = %g\n",dblrtn);

    return (i);
    }
```

A.1.2. Creating a Shared Library

Once you have written the source code for your functions, you can create a Makefile (using the **gen_share** utility) to build a shared library. The utility creates the Makefile in the current directory. The Makefile incorporates all the interdependencies of the C source files it encounters in that current directory. The **gen_share** utility is meant to setup the basic build. The user may need to make modifications to the Makefile depending on the situation.

The **gen_share** utility has the following syntax:

```
gen_share [-h] [-64] shared_object_name
```

where

-h

Produces command help.

-64

Configures the Makefile to use the **-mips4** option for IRIX64 .

shared_object_name

Is the name that will be given to the shared library.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

As gen_share is executing, you may see one or more "No match" messages. This is normal. The script is searching for .c, .f, and .F file types in the current directory and returns this message if it cannot locate any files matching one of those types.

To create a shared library called mylibrary.so, you would issue the following command:

```
% gen_share mylibrary.so
```

The utility will produce a Makefile in the current directory. You will be able to generate the shared library by issuing the following command:

```
make
```

For example, to create the shared library for mylibrary.so, you would issue the following command:

```
% make
```

You will then find the specified shared library file in the current directory. You may also see warnings from the make process, and you may need to modify the Makefile or your source code.

A.1.3. Creating an External Table File

The external table file (ans_ext.tbl) can reside in any directory (but you must specify that directory in the **ANSYS_EXTERNAL_PATH** environment variable). The file contains an entry for each shared library function you wish to allow Mechanical APDL to access. There is no limit to the number of entries. The file entries have the following format:

/shared/library/path/library.so *~cm_name function_name*

where:

/shared/library/path/library.so is the path to the directory that contains the shared library file. (Remotely mounted file systems are not recommended.)

~cm_name is the command used to invoke the function within Mechanical APDL. The command name must begin with a tilde (~) and each command name must be unique within the first four characters. The command name must be eight characters or less, including the tilde (~).

function_name is the name of the function that is referenced by the specified command name. (This must be unique within the first four characters if multiple external commands are specified.)

For example, the following entry references the /home/mydir/mylibs/myobject.so shared library and the *myobject_* function. It specifies *~myobj* as the related command:

```
/home/mydir/mylibs/myobject.so ~myobj myobject_
```

Mechanical APDL also makes use of external commands, and places its own shared libraries and the associated external table file in the /ansys_inc/v212/ansys/lib/<*platform*> directory (where <*platform*> is the directory specific to your computing platform, such as /linx64).

Mechanical APDL loads external commands as follows:

- Checks the ans_ext.tbl file in the /ansys_inc/v212/ansys/lib/<*platform*> directory and loads any external commands referenced there.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Loads external commands referenced by the external table file in the directory designated with the **ANSYS_EXTERNAL_PATH** environment variable (see section Setting the ANSYS_EXTERNAL_PATH Environment Variable (p. 386)).

If you designate a command name that has the same first four characters as a command listed in the `/ansys_inc/v212/ansys/lib/<platform>/ans_ext.tbl` file, you will not be able to access your command. Therefore, it is a good practice to check the external table file to make sure you have no external command name conflicts. *Do not* modify the `/ansys_inc/v212/ansys/lib/<platform>/ans_ext.tbl` file. You can also use the **~DEBUG** command to verify that no external command name conflicts exist.

Note:

The shared library must be consistent with the computer type and OS level on which Mechanical APDL is executed.

A.1.4. Setting the ANSYS_EXTERNAL_PATH Environment Variable

Before launching Mechanical APDL, first set the **ANSYS_EXTERNAL_PATH** to point to the directory containing the external table file. (For convenience, if you distribute your new functionality to other users they should set their `.login` or `.cshrc` files so that it is persistent from session to session.) For example, the following sets the environment variable to point to the `/home/mydir` directory.

```
setenv ANSYS_EXTERNAL_PATH /home/mydir
```

A.1.5. Using External Commands

To call an external command, enter it as you would any other Mechanical APDL command. You can also call external commands through either an APDL macro or UIDL script.

Note:

Avoid recursive external commands; that is, avoid situations where an external command calls another external command.

A.1.6. Checking External Command Status

You can check what shared libraries are currently accessible by entering the **~DEBUG** command in the command input window. The following figure shows an example of **~DEBUG** command output.

```
External Command Mappings:

 Command   Library                              Function         Accessed?
  *-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-
 ~excmd    /home/mydir/mycode/mycommand.so      excmd            YES
```

In this example, the output lists the command, the related shared library, the function, and if the command has been accessed.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A.1.7. Resetting External Commands

You can

- Close all shared libraries
- Free memory associated with external commands

by issuing the **~RESET** command. The command issues the following message to confirm that the reset operation was complete.

```
~RESET was processed: The external command buffers have been cleared.
```

Note:

The **/CLEAR** command also closes/resets all external command shared libraries.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Appendix B. Creating External Commands in Windows

This section describes the steps required to create external commands on Windows platforms.

B.1. Tasks in Creating an External Command

To create a functional external command, you will need to complete the following general steps:

- Create compatible C source code.
- Create an external definition file (`projname.def`).
- Create a new project in Microsoft Visual Studio 2017.
- Create a shared library.
- Create an external table file (`ans_ext.tbl`), listing the various shared libraries, each function and the related command.
- Set the **ANSYS_EXTERNAL_PATH** environment variable

The following sections detail each of these tasks.

B.1.1. Creating Compatible Code

You can create your functions using any of the API functions described in `Program Files\ANSYS Inc\V212\ansys\customize\include\cAnsInterface.h`, `cAnsQuery.h`, and `cAnspick.h`. You can then execute these functions via the "external command" feature within Mechanical APDL. In addition, the program provides special commands that list all available external commands and allow you to reset all currently referenced external commands. The following code segment demonstrates, at a minimal level, how to create functions that can be used as an entry point into a custom coded shared library.

The most important point in the following example is:

- The C program interface is an integer function that has one argument (a char pointer).

```
#include <windows.h>
#include "cAnsInterface.h"
#include "CAnsQuery.h"

/*
------------------------------  Function Description -----------------------
extfnc
  int extfnc(uecmd)
  char *uecmd;

Purpose:

     Demonstrate C API entry function for an external command.
```

```
Parameters:

     Input
     ----------------------------
       uecmd
         The ANSYS external command string.

     Output
     ----------------------------

Return Value:
     The return value is ignored by the calling function;

---------------------------- End Function  Description --------------------

*/
int extfnc(char* uecmd)
{
   /* Note: uecmd is the entire command given to invoke this function */
    char* cmdsend = {"/COM, COMMAND SENT FROM EXTERNAL COMMAND"};
    char* querystr = {"NODE,,NUM,MAX"};
        char strrtn[32];
    int i, itype;
    double dblrtn;

    /* Send a simple command to be executed */
    i = cAnsSendCommand(cmdsend);

    /* Perform a simple query */
    i = cAnsGetValue(querystr,&dblrtn,strrtn,&itype);

    /* Display the value retrieved */
    cAns_printf("Max Node Number = %g\n",dblrtn);

    return (i);
    }
```

B.1.2. Creating a Visual Studio Project

The steps for building a Visual Studio project are demonstrated in the example at the end of this appendix. See Example: Creating an External Command Using Visual Studio 2017 Professional (p. 393).

B.1.3. Creating an External Definition File

For each external function, you must declare it in the external definition file. The naming convention for this file is the name of your project with the `.def` extension; it must be located in your project directory. This file consists of the word *EXPORTS* on the first line, and the name(s) of the functions to be exported on each successive line. For the example function above:

```
EXPORTS


extfunc
```

B.1.4. Creating a Shared Library

Once all of the necessary files have been incorporated into your project, simply compile (**Ctrl**+**F7**) and build (**F7**) the project. In your project directory, Developer Studio will create a Debug directory and will place the library in that directory (`projname.dll`).

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

B.1.5. Creating an External Table File

The external table file (`ans_ext.tbl`) can reside in any directory (but you must specify that directory in the **ANSYS_EXTERNAL_PATH** environment variable). The file contains an entry for each shared library function you wish Mechanical APDL to access. There is no limit to the number of entries. The file entries have the following format:

`C:\shared\library\path\projname.dll ~cm_name function_name`

where:

`C:\shared\library\path\projname.dll` is the path to the directory that contains the shared library file. (Remotely mounted file systems are not recommended.)

~cm_name is the command used to invoke the function within Mechanical APDL. The command name must begin with a tilde (~) and the first four characters of each command name must be unique.

function_name is the name of the function that is referenced by the specified command name. (This must be unique within the first four characters if multiple external commands are specified.)

For example, the following entry references the `C:\home\mydir\mylibs\myobject.dll` shared library and the *myobject* function, and specifies *~myobj* as the related command:

```
C:\home\mydir\mylibs\myobject.dll ~myobj myobject
```

Mechanical APDL also makes use of external commands, and places its own shared libraries and the associated external table file in the `C:\Program Files\ANSYS Inc\V212\ansys\lib\<platform>` directory (where *<platform>* is the directory specific to your computing platform, such as `\winx64`). The program loads external commands in the following order:

- Checks the `ans_ext.tbl` file in the `C:\Program Files\ANSYS Inc\V212\ansys\lib\<platform>` directory and loads any external commands referenced there.
- Loads external commands referenced by the external table file in the directory designated with the **ANSYS_EXTERNAL_PATH** environment variable (see Setting the ANSYS_EXTERNAL_PATH Environment Variable (p. 392)).

If you designate a command name that has the same first four characters as a command listed in the `C:\Program Files\ANSYS Inc\V212\ansys\lib\<platform>\ans_ext.tbl` file, you will not be able to access your command. Therefore, it is a good practice to check the external table file to make sure you have no external command name conflicts. Do not modify the `C:\Program Files\ANSYS Inc\V212\ansys\lib\<platform>\ans_ext.tbl` file. You can also use the **~DEBUG** command to verify that no external command name conflicts exist.

Note:

The shared library must be consistent with the computer type and OS level on which Mechanical APDL is executed.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

B.1.6. Setting the ANSYS_EXTERNAL_PATH Environment Variable

Before launching Mechanical APDL, you must first set the **ANSYS_EXTERNAL_PATH** to point to the directory containing the external table file. In Windows NT, the environment variables are in System Properties, which can be accessed through the Control Panel. For example, the following string sets the environment variable to point to the `C:\home\mydir` directory.

```
set ANSYS_EXTERNAL_PATH=C:\home\mydir
```

B.1.7. Using External Commands

To call an external command, issue it as you would any other Mechanical APDL command. You can also call external commands through either an APDL macro or UIDL routine.

Note:

Avoid recursive external commands; that is, avoid situations where an external command calls another external command.

B.1.8. Checking External Command Status

You can check what shared libraries are currently accessible by entering the **~DEBUG** command in the command input window. The following figure shows an example of **~DEBUG** command output.

```
External Command Mappings:
 Command   Library                            Function         Accessed?
  *-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-*-
 ~excmd    /home/mydir/mycode/mycommand.so     excmd            YES
```

Note that the output lists the command, the related shared library, the function, and whether or not the command has been accessed.

B.1.9. Resetting External Commands

You can

- Close all shared libraries
- Free memory associated with external commands

by issuing the **~RESET** command. This command issues the following message to confirm that the reset operation is complete.

```
~RESET was processed: The external command buffers have been cleared.
```

Note:

The **/CLEAR** command also closes/resets all external command shared libraries.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

B.1.10. Example: Creating an External Command Using Visual Studio 2017 Professional

An example for setting up an external command using Microsoft Visual Studio 2017 Professional is provided on the installation media. To run this example, perform the following steps.

1. Go to the `Program Files\ANSYS Inc\V212\ANSYS\custom\user\winx64\ExtCmd` directory.
2. Open the Visual Studio 2017 Professional solution file `extcmd.sln` (double click the file).
3. From the Visual Studio 2017 Professional menu, click on `Build->Rebuild Solution`.
4. Exit Visual Studio 2017 Professional.
5. Double-click on `runextcmdtest.bat` to run Mechanical APDL and test the external command that was just compiled.
6. In the output window enter `~excmd`. You should see the following:

```
BEGIN:
~excmd
  COMMAND SENT FROM EXTERNAL COMMAND
 Max Node Number = 0

 *** NOTE ***                            CP =       0.625   TIME= 14:24:33
 Command= ~excmd was processed as an external command which is a
 non-standard use of the Mechanical APDL program.
```

7. Enter `/exit,nosave` to exit the program.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Appendix C. User Material (UserMat) Subroutine Example

This example of a simple bilinear plasticity material model (identical to **TB**,PLAS,,,,BISO) demonstrates the user material subroutine `UserMat`, described in Subroutine UserMat (Creating Your Own Material Model) (p. 205).

C.1. UserMat Example Description

The example subroutine defines a 3-D material with plane strain and axisymmetric stress states. The analysis uses the 3-D solid element SOLID185. Comparison is made with the prediction by the **TB**,PLAS,,,,BISO material option.

The example is a two-element test case under simple tension. Element 1 has material defined using the **TB**,USER option, while Element 2 has material defined using the **TB**,PLAS,,,,BISO option. A 100-percent deformation is applied to both elements. Finite deformation (**NLGEOM**, ON) is considered. The POST26 processor results of stress components (S*xx*, S*yy*) and plastic strain components (EP*xx*, EP*yy*) are printed for both elements. They are expected to be the same.

C.2. UserMat Example Input Data

```
/batch,list
/title, mvpl-um01, gal, usermat.F test case
/com,
/com, This is a single element test case for testing usermat.F
/com, usermat.F is the user materials subroutine
/com  for current-technology elements.
/com, The material subroutine provided as the example
/com, is the same as the TB,PLAS,,,,BISO.
/com, A side by side comparison is made for two 185 elements,
/com, among which one is defined by TB,PLAS,,,,BISO, and another
/com, is defined as TB,USER. They are expected to produce
/com, the same results.
/com, uniaxial tension stress, large deformation.
/com,
/nopr
/nolist
/prep7

ele1=185
ele2=185
mat1=1
mat2=2


et,1,ele1
keyopt,1,2,1
mat,mat1
block,0,1,0,1,0,1
esize,,1
vmesh,1

mat,mat2
block,0,1,0,1,0,1
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
esize,,1
vmesh,2

elist

! define material 1 by tb,plas,,,,biso

EX=20e5
ET=100
EP=EX*ET/(EX-ET)
mp,ex  ,mat1,EX
mp,nuxy,mat1,0.3
tb,plas,mat1,,,biso
tbtemp,1.0
tbdata,1,1e3,EP
tbtemp,2.0
tbdata,1,2e3,EP

! define material 2 by tb,user

tb,user,mat2,2,4
tbtemp,1.0                          ! first temp.
tbdata,1,19e5, 0.3, 1e3,100,       ! E, posn, sigy, H
tbtemp,2.0
tbdata,1,21e5, 0.3, 2e3,100,
tb,state,mat2,,8                    ! define 8 state variables

! boundary condition

nsel,s,loc,x
d,all,ux
nall
nsel,s,loc,y
d,all,uy
nall
nsel,s,loc,z
d,all,uz
nall
fini

/solu
tunif,1.5
nlgeom,on
nsel,s,loc,y,1
nsubst,20,100,1
d,all,uy,1.0
time,1
nall
outres,,-10
outpr,all,-10
solv

fini
/post26
esol,2,1,,s,x,SX_BISO
esol,3,2,,s,x,SX_USER
esol,4,1,,s,y,SY_BISO
esol,5,2,,s,y,SY_USER
esol,6,1,,eppl,x,EPX_BISO
esol,7,2,,eppl,x,EPX_USER
esol,8,1,,eppl,y,EPY_BISO
esol,9,2,,eppl,y,EPY_USER

prvar,2,3,4,5
prvar,6,7,8,9

fini

/exit,no save
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

C.3. UserMat Example POST26 Output Results

```
          ***** ANSYS POST26 VARIABLE LISTING *****


    TIME          1 S   X       2 S   X       1 S   Y       2 S   Y
                  SX_BISO       SX_USER       SY_BISO       SY_USER
  0.10000       -0.188102E-02 -0.188102E-02   1509.45       1509.45
  0.28750       -0.110968     -0.110968       1525.07       1525.07
  0.45625       -0.814415     -0.814415       1536.67       1536.67
  0.66204        -1.73160      -1.73160       1548.95       1548.95
  0.89592        -1.86240      -1.86240       1561.97       1561.97
   1.0000       -0.176924E-01 -0.176924E-01   1569.16       1569.16


          ***** ANSYS POST26 VARIABLE LISTING *****


    TIME          1 EPPLX       2 EPPLX       1 EPPLY       2 EPPLY
                  EPX_BISO      EPX_USER      EPY_BISO      EPY_USER
  0.10000       -0.472687E-01 -0.472687E-01  0.945374E-01  0.945374E-01
  0.28750       -0.125917     -0.125917      0.251834      0.251834
  0.45625       -0.187417     -0.187417      0.374835      0.374835
  0.66204       -0.253409     -0.253409      0.506818      0.506818
  0.89592       -0.319141     -0.319141      0.638282      0.638282
   1.0000       -0.345853     -0.345853      0.691707      0.691707
```

C.4. USERMAT.F List File for This Example

```
      subroutine usermat(
     &                   matId, elemId,kDomIntPt, kLayer, kSectPt,
     &                   ldstep,isubst,keycut,
     &                   nDirect,nShear,ncomp,nStatev,nProp,
     &                   Time,dTime,Temp,dTemp,
     &                   stress,ustatev,dsdePl,sedEl,sedPl,epseq,
     &                   Strain,dStrain, epsPl, prop, coords,
     &                   var0, defGrad_t, defGrad,
     &                   tsstif, epsZZ,
     &                   var1, var2, var3, var4, var5,
     &                   var6, var7, var8)
c*************************************************************************
c     *** primary function ***
c
c           user defined material constitutive model
c
c      Attention:
c           User must define material constitutive law properly
c           according to the stress state such as 3D, plane strain
c           and axisymmetry, plane stress and 3D/1D beam.
c
c           a 3D material constitutive model can use for
c           plane strain and axisymmetry cases.
c
c           When using shell elements, a plane stress algorithm
c           must be use.
c
c
c       The following demonstrates a USERMAT subroutine for
c       a plasticity model, which is the same as TB, BISO,
c       for different stress states.
c       See "ANSYS user material subroutine USERMAT" for detailed
c       description of how to write a USERMAT routine.
c
c       This routine calls four routines,
c       usermat3d.F, usermatps.F usermatbm.F and usermat1d.F, w.r.t.
c       the corresponding stress states.
c       Each routine can be also a usermat routine for the specific
```

```
c       element.
c
c*************************************************************************
c
c     input arguments
c     ===============
c      matId     (int,sc,i)              material #
c      elemId    (int,sc,i)              element #
c      kDomIntPt (int,sc,i)              "k"th domain integration point
c      kLayer    (int,sc,i)              "k"th layer
c      kSectPt   (int,sc,i)              "k"th Section point
c      ldstep    (int,sc,i)              load step number
c      isubst    (int,sc,i)              substep number
c      nDirect   (int,sc,in)             # of direct components
c      nShear    (int,sc,in)             # of shear components
c      ncomp     (int,sc,in)             nDirect + nShear
c      nstatev   (int,sc,l)              Number of state variables
c      nProp     (int,sc,l)              Number of material ocnstants
c
c      Temp      (dp,sc,in)              temperature at beginning of
c                                        time increment
c      dTemp     (dp,sc,in)              temperature increment
c      Time      (dp,sc,in)              time at beginning of increment (t)
c      dTime     (dp,sc,in)              current time increment (dt)
c
c      Strain   (dp,ar(ncomp),i)         Strain at beginning of time increment
c      dStrain  (dp,ar(ncomp),i)         Strain increment
c      prop     (dp,ar(nprop),i)         Material constants defined by TB,USER
c      coords   (dp,ar(3),i)             current coordinates
c      defGrad_t(dp,ar(3,3),i)           Deformation gradient at time t
c      defGrad  (dp,ar(3,3),i)           Deformation gradient at time t+dt
c
c     input output arguments
c     ======================
c      stress   (dp,ar(nTesn),io)        stress
c      ustatev   (dp,ar(nstatev),io)     user state variables
c      sedEl    (dp,sc,io)               elastic work
c      sedPl    (dp,sc,io)               plastic work
c      epseq    (dp,sc,io)               equivalent plastic strain
c      tsstif   (dp,ar(2),io)            transverse shear stiffness
c                                        tsstif(1) - Gxz
c                                        tsstif(2) - Gyz
c                                        tsstif(1) is also used to calculate hourglass
c                                        stiffness, this value must be defined when low
c                                        order element, such as 181, 182, 185 with uniform
c                                        integration is used.
c      var?     (dp,sc,io)               not used, they are reserved arguments
c                                        for further development
c
c     output arguments
c     ================
c      keycut   (int,sc,io)              loading bisect/cut control
c                                        0 - no bisect/cut
c                                        1 - bisect/cut
c                                        (factor will be determined by ANSYS solution control)
c      dsdePl   (dp,ar(ncomp,ncomp),io)  material jacobian matrix
c      epsZZ    (dp,sc,o)                strain epsZZ for plane stress,
c                                        define it when accounting for thickness change
c                                        in shell and plane stress states
c
c*************************************************************************
c
c      ncomp   6   for 3D  (nshear=3)
c      ncomp   4   for plane strain or axisymmetric (nShear = 1)
c      ncomp   3   for plane stress (nShear = 1)
c      ncomp   3   for 3d beam      (nShear = 2)
c      ncomp   1   for 1D (nShear = 0)
c
c      stresss and strains, plastic strain vectors
c          11, 22, 33, 12, 23, 13    for 3D
c          11, 22, 33, 12            for plane strain or axisymmetry
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

```
c          11, 22, 12               for plane stress
c          11, 13, 12               for 3d beam
c          11                       for 1D
c
c      material jacobian matrix
c        3D
c           dsdePl    |  1111   1122   1133   1112   1123   1113 |
c           dsdePl    |  2211   2222   2233   2212   2223   2213 |
c           dsdePl    |  3311   3322   3333   3312   3323   3313 |
c           dsdePl    |  1211   1222   1233   1212   1223   1213 |
c           dsdePl    |  2311   2322   2333   2312   2323   2313 |
c           dsdePl    |  1311   1322   1333   1312   1323   1313 |
c        plane strain or axisymmetric (11, 22, 33, 12)
c           dsdePl    |  1111   1122   1133   1112 |
c           dsdePl    |  2211   2222   2233   2212 |
c           dsdePl    |  3311   3322   3333   3312 |
c           dsdePl    |  1211   1222   1233   1212 |
c        plane stress (11, 22, 12)
c           dsdePl    |  1111   1122   1112 |
c           dsdePl    |  2211   2222   2212 |
c           dsdePl    |  1211   1222   1212 |
c        3d beam (11, 13, 12)
c           dsdePl    |  1111   1113   1112 |
c           dsdePl    |  1311   1313   1312 |
c           dsdePl    |  1211   1213   1212 |
c        1d
c           dsdePl    |  1111 |
c
c**********************************************************************
#include "impcom.inc"
c
      INTEGER
     &                 matId, elemId,
     &                 kDomIntPt, kLayer, kSectPt,
     &                 ldstep,isubst,keycut,
     &                 nDirect,nShear,ncomp,nStatev,nProp
      DOUBLE PRECISION
     &                 Time,    dTime,   Temp,    dTemp,
     &                 sedEl,   sedPl,   epseq,   epsZZ
      DOUBLE PRECISION
     &                 stress  (ncomp  ), ustatev  (nStatev),
     &                 dsdePl  (ncomp,ncomp),
     &                 Strain  (ncomp  ), dStrain (ncomp  ),
     &                 epsPl   (ncomp  ), prop    (nProp  ),
     &                 coords  (3),
     &                 defGrad_t(3,3),    defGrad(3,3),
     &                 tsstif  (2)
c
c***************** User defined part **********************************
c
c --- parameters
c
      INTEGER          NEWTON, mcomp
      DOUBLE PRECISION HALF, THIRD, ONE, TWO, SMALL, ONEHALF,
     &                 ZERO, TWOTHIRD, ONEDM02, ONEDM05, sqTiny
      PARAMETER       (ZERO       = 0.d0,
     &                 HALF       = 0.5d0,
     &                 THIRD      = 1.d0/3.d0,
     &                 ONE        = 1.d0,
     &                 TWO        = 2.d0,
     &                 SMALL      = 1.d-08,
     &                 sqTiny     = 1.d-20,
     &                 ONEDM02    = 1.d-02,
     &                 ONEDM05    = 1.d-05,
     &                 ONEHALF    = 1.5d0,
     &                 TWOTHIRD   = 2.0d0/3.0d0,
     &                 NEWTON     = 10,
     &                 mcomp      = 6
     &                 )
c
c --- local variables
```

```
c
c      sigElp   (dp,ar(6  ),l)           trial stress
c      dsdeEl   (dp,ar(6,6),l)           elastic moduli
c      sigDev   (dp,ar(6  ),l)           deviatoric stress tensor
c      dfds     (dp,ar(6  ),l)           derivative of the yield function
c      JM       (dp,ar(6,6),l)           2D matrix for a 4 order tensor
c      pEl      (dp,sc     ,l)           hydrostatic pressure stress
c      qEl      (dp,sc     ,l)           von-mises stress
c      pleq_t   (dp,sc     ,l)           equivalent plastic strain at
c                                        beginnig of time increment
c      pleq     (dp,sc     ,l)           equivalent plastic strain at end
c                                        of time increment
c      dpleq    (dp,sc     ,l)           incremental equivalent plastic strain
c      cpleq    (dp,sc     ,l)           correction of incremental
c                                        equivalent plastic strain
c      sigy_t   (dp,sc     ,l)           yield stress at beginnig of time increments
c      sigy     (dp,sc     ,l)           yield stress at end of time
c                                        increment
c      young    (dp,sc     ,l)           Young's modulus
c      posn     (dp,sc     ,l)           Poiss's ratio
c      sigy0    (dp,sc     ,l)           initial yield stress
c      dsigdep  (dp,sc     ,l)           plastic slope
c      twoG     (dp,sc     ,l)           two time of shear moduli
c      threeG   (dp,sc     ,l)           three time of shear moduli
c      funcf    (dp,sc     ,l)           nonlinear function to be solved
c                                        for dpleq
c      dFdep    (dp,sc     ,l)           derivative of nonlinear function
c                                        over dpleq
c
c --- temporary variables for solution purpose
c      i, j
c      threeOv2qEl, oneOv3G, qElOv3G, con1, con2, fratio
c
       DOUBLE PRECISION sigElp(mcomp), dsdeEl(mcomp,mcomp), G(mcomp),
      &                 sigDev(mcomp), JM    (mcomp,mcomp), dfds(mcomp)

       DOUBLE PRECISION var0, var1, var2, var3, var4, var5,
      &                 var6, var7, var8

       DATA G/1.0D0,1.0D0,1.0D0,0.0D0,0.0D0,0.0D0/
c
       INTEGER          i, j
       DOUBLE PRECISION pEl,   qEl,     pleq_t,  sigy_t , sigy,
      &                 cpleq, dpleq,   pleq,
      &                 young, posn,    sigy0,   dsigdep,
      &                 elast1,elast2,
      &                 twoG,  threeG,  oneOv3G, qElOv3G, threeOv2qEl,
      &                 funcf, dFdep,   fratio,  con1,    con2
c*************************************************************************
c
       keycut   = 0
       dsigdep  = ZERO
       pleq_t   = ustatev(1)
       pleq     = pleq_t
c *** get Young's modulus and Poisson's ratio, initial yield stress and others
       young    = prop(1)
       posn     = prop(2)
       sigy0    = prop(3)
c *** calculate the plastic slope
       dsigdep  = young*prop(4)/(young-prop(4))
       twoG     = young / (ONE+posn)
       threeG   = ONEHALF * twoG
c
c *** calculate elastic stiffness matrix (3-D)
c
c
       elast1=young*posn/((1.0D0+posn)*(1.0D0-TWO*posn))
       elast2=young/(TWO*(1.0D0+posn))
       dsdeEl(1,1)=(elast1+TWO*elast2)*G(1)*G(1)
       dsdeEl(1,2)=elast1*G(1)*G(2)+elast2*TWO*G(4)*G(4)
       dsdeEl(1,3)=elast1*G(1)*G(3)+elast2*TWO*G(5)*G(5)
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
      dsdeEl(1,4)=elast1*G(1)*G(4)+elast2*TWO*G(1)*G(4)
      dsdeEl(1,5)=elast1*G(1)*G(5)+elast2*TWO*G(1)*G(5)
      dsdeEl(1,6)=elast1*G(1)*G(6)+elast2*TWO*G(4)*G(5)
      dsdeEl(2,2)=(elast1+TWO*elast2)*G(2)*G(2)
      dsdeEl(2,3)=elast1*G(2)*G(3)+elast2*TWO*G(6)*G(6)
      dsdeEl(2,4)=elast1*G(2)*G(4)+elast2*TWO*G(1)*G(4)
      dsdeEl(2,5)=elast1*G(2)*G(5)+elast2*TWO*G(1)*G(5)
      dsdeEl(2,6)=elast1*G(2)*G(6)+elast2*TWO*G(2)*G(6)
      dsdeEl(3,3)=(elast1+TWO*elast2)*G(3)*G(3)
      dsdeEl(3,4)=elast1*G(3)*G(4)+elast2*TWO*G(5)*G(6)
      dsdeEl(3,5)=elast1*G(3)*G(5)+elast2*TWO*G(5)*G(3)
      dsdeEl(3,6)=elast1*G(3)*G(6)+elast2*TWO*G(6)*G(3)
      dsdeEl(4,4)=elast1*G(4)*G(4)+elast2*(G(1)*G(2)+G(4)*G(4))
      dsdeEl(4,5)=elast1*G(4)*G(5)+elast2*(G(1)*G(6)+G(5)*G(4))
      dsdeEl(4,6)=elast1*G(4)*G(6)+elast2*(G(4)*G(6)+G(5)*G(2))
      dsdeEl(5,5)=elast1*G(5)*G(5)+elast2*(G(1)*G(3)+G(5)*G(5))
      dsdeEl(5,6)=elast1*G(5)*G(6)+elast2*(G(4)*G(3)+G(5)*G(6))
      dsdeEl(6,6)=elast1*G(6)*G(6)+elast2*(G(2)*G(3)+G(6)*G(6))
      do i=1,ncomp-1
        do j=i+1,ncomp
          dsdeEl(j,i)=dsdeEl(i,j)
        end do
      end do
c
c *** calculate the trial stress and
c     copy elastic moduli dsdeEl to material Jacobian matrix
      do i=1,ncomp
         sigElp(i) = stress(i)
         do j=1,ncomp
            dsdePl(j,i) = dsdeEl(j,i)
            sigElp(i) = sigElp(i)+dsdeEl(j,i)*dStrain(j)
         end do
      end do
c *** hydrostatic pressure stress
      pEl = -THIRD * (sigElp(1) + sigElp(2) + sigElp(3))
c *** compute the deviatoric stress tensor
      sigDev(1) = sigElp(1) + pEl
      sigDev(2) = sigElp(2) + pEl
      sigDev(3) = sigElp(3) + pEl
      sigDev(4) = sigElp(4)
      sigDev(5) = sigElp(5)
      sigDev(6) = sigElp(6)
c *** compute von-mises stress
      qEl =
     &  sigDev(1) * sigDev(1)+sigDev(2) * sigDev(2)+
     &  sigDev(3) * sigDev(3)+
     &  TWO*(sigDev(4) * sigDev(4)+ sigDev(5) * sigDev(5)+
     &  sigDev(6) * sigDev(6))
      qEl = sqrt( ONEHALF * qEl)
c *** compute current yield stress
      sigy    = sigy0 + dsigdep * pleq
c
      fratio = qEl / sigy - ONE
c *** check for yielding
      IF (sigy .LE. ZERO.or.fratio .LE. -SMALL) GO TO 500
c
      sigy_t = sigy
      threeOv2qEl = ONEHALF / qEl
c *** compute derivative of the yield function
      DO i=1, ncomp
         dfds(i) = threeOv2qEl * sigDev(i)
      END DO
      oneOv3G  = ONE / threeG
      qElOv3G  = qEl * oneOv3G
c *** initial guess of incremental equivalent plastic strain
      dpleq    = (qEl - sigy) * oneOv3G
      pleq     = pleq_t + dpleq
c
c *** Newton-Raphson procedure for return mapping iteration
      DO i = 1,NEWTON
         sigy   = sigy0 + dsigdep * pleq
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
         funcf  = qElOv3G -  dpleq - sigy * oneOv3G
         dFdep  = - ONE  - dsigdep * oneOv3G
         cpleq  = -funcf / dFdep
         dpleq  = dpleq + cpleq
c        --- avoid negative equivalent plastic strain
         dpleq  = max (dpleq, sqTiny)
         pleq   = pleq_t + dpleq
         fratio = funcf/qElOv3G
c ***    check covergence
         IF (((abs(fratio) .LT. ONEDM05        )  .AND.
     &        (abs(cpleq ) .LT. ONEDM02 * dpleq)) .OR.
     &       ((abs(fratio) .LT. ONEDM05        )  .AND.
     &        (abs(dpleq ) .LE. sqTiny         )))  GO TO 100
      END DO
c
c *** Uncovergence, set keycut to 1 for bisect/cut
      keycut   = 1
      GO TO 990
 100  CONTINUE
c
c ***  update stresses
      con1 =  twoG * dpleq
      DO i = 1 , ncomp
         stress(i) =  sigElp(i) - con1 * dfds(i)
      END DO
c
c ***  update plastic strains
      DO i = 1 , nDirect
         epsPl(i) = epsPl(i) + dfds(i) * dpleq
      END DO
      DO i = nDirect + 1 , ncomp
         epsPl(i) = epsPl(i) + TWO * dfds(i) * dpleq
      END DO
      epseq  = pleq
c *** Update state variables
      ustatev(1) = pleq
c *** Update plastic work
      sedPl = sedPl + HALF * (sigy_t+sigy)*dpleq
c
c *** Material Jcobian matrix
c
      IF (qEl.LT.sqTiny) THEN
         con1 = ZERO
      ELSE
         con1 = threeG * dpleq / qEl
      END IF
      con2 = threeG/(threeG+dsigdep) - con1
      con2 = TWOTHIRD * con2
      DO i=1,ncomp
         DO j=1,ncomp
            JM(j,i) = ZERO
         END DO
      END DO
      DO i=1,nDirect
         DO j=1,nDirect
            JM(i,j) = -THIRD
         END DO
         JM(i,i) = JM(i,i) + ONE
      END DO
      DO i=nDirect + 1,ncomp
         JM(i,i) = HALF
      END DO
      DO i=1,ncomp
         DO j=1,ncomp
            dsdePl(i,j) =    dsdeEl(i,j) - twoG
     &           * (  con2 * dfds(i) * dfds(j) + con1 * JM(i,j) )
         END DO
      END DO
c
      goto 600
  500 continue
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
c *** Update stress in case of elastic/unloading
      do i=1,ncomp
         stress(i) = sigElp(i)
      end do

  600 continue
c *** Claculate elastic work
      sedEl = ZERO
      DO i = 1 , ncomp
         sedEl = sedEl + stress(i)*(Strain(i)+dStrain(i)-epsPl(i))
      END DO
      sedEl    = sedEl * HALF
c
 990  CONTINUE
c
      return
      end
```

C.5. Accessing Solution and Material Data

These APIs are provided for your convenience to help you access solution and material data easily.

```
c *** subroutine get_ElmInfo(inquire, value)
c
c        description
c           function to inquire element and solution information
c        definition
c           inquire - query argument (string)
c           value   - value of query argument
c        variables
c           inquire        - value
c           'LDSTEP'       - load step number
c           'ISUBST'       - substep step number
c           'IEQITR'       - current interation number
c           'NUMINTG'      - number of gauss integration
c           'ELEMID'       - element number
c           'MATID'        - material number of current element
c           'NSVAR'        - number of state variable for current material at
c                            gauss intg.
c           'NCOMP'        - number of vector components, such as stresses
c

c *** subroutine get_ElmData (kchar, elemId, kMatRecPt, ncomp, vect)
c
c        description
c           function to get/inquire solution dependent variables
c           such as stress, strains at gauss intg. point.
c        definition
c           kchar          - string variable containing a query argument
c           elemId         - element number
c           kMatRecPt      - material integration point number
c           ncomp          - number of components to be inquired
c                               Use the 'NCOMP' query to determine the correct
c                               array sizes for tensor quantities
c           vect(*)        - variable array containing the retrieved variables
c        variables
c          'ESYS'          - element coordinate systems
c          'ISIG'          - initial stress
c          'SIG'           - stress (for CPT elements with pore pressure DOFs, this query returns effective stress)
c          'EPTO'          - total strain
c          'EPPL'          - plastic strain
c          'EPCR'          - creep strain
c          'EPTH'          - thermal strain
c          'EPSW'          - swelling strain
c          'SVAR'          - state variables
c          'PLEQ'          - accumulated equivalent plastic strain
```

```
c          'CREQ'          - accumulated equivalent creep strain
c          'TSIG'          - total stress (this is available only for CPT elements with pore pressure)

c
c *** subroutine put_ElmData (inquire, elemId, kIntg, nvect, vect)
c        description
c           function to put solution dependent variables
c           such as stress, strains at gauss intg. point.
c        !! Use this in caution, it overides ansys database. Usually
c        !! you should only write user defined state variables,
c        !! SVAR
c        definition
c           inquire        - query argument (string)
c           elemId         - element number
c           kIntg          - gauss intg. number
c           nvect          - number of vector to be inquired
c                              Use the 'NCOMP' query to determine the correct
c                              array sizes for tensor quantities
c           vect           - vector to be inquired
c
c        variables
c          'SIG '          - stress vector
c          'EPTO'          - Total   strain vector (EPEL+EPPL+EPCR+EPTH)
c          'EPPL'          - plastic strain vector
c          'EPCR'          - creep   strain vector
c          'EPTH'          - thermal strain vector
c          'ISIG'          - Initial stress vector
c          'PLEQ'          - accumulated equivalent plastic strain
c          'CREQ'          - accumulated equivalent creep   strain
c          'SVAR'          - State variables (define by tb,state)
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Appendix D. Structural-Thermal User Material (UserMat, UserMatTh) Example

This example demonstrates user-defined structural and thermal materials in a coupled structural-thermal analysis.

D.1. Example Description

The problem considered in the example is a 3-D version of the verification test case VM296 in the *Ansys Mechanical APDL Verification Manual*.

The bilinear isotropic hardening plastic material with temperature-dependent yield stress and tangent modulus is defined using the **TB**,USER model. The linear thermal material is defined using **TB**,USER,,,THERM. The source files for the `UserMat.F` and `UserMatTH.F` subroutines used in this example can be found in subdirectory: `/ansys_inc/v212/ansys/customize/user/` (Linux) or `Program Files\ANSYS Inc\V212\ansys\customize\user\` (Windows).

The model is discretized using SOLID226 elements.

Results are expected to agree closely with the results of a 2-D simulation (VM296) performed for simulation time = 13 s.

D.2. Example Input Data

```
/TITLE,USER MATERIAL FOR RADIAL EXPANSION OF A THERMOPLASTIC CYLINDER
/COM,
/COM, REFERENCE: "THERMOMECHANICAL MODELING OF METALS AT FINITE STRAINS:
/COM, FIRST AND MIXED ORDER FINITE ELEMENTS", LAURENT ADAM, JEAN-PHILIPPE PONTHOT,
/COM, INTERNATIONAL JOURNAL OF SOLIDS AND STRUCTURES 42 (2005) 5615-5655
/COM,
RI=0.1               ! INTERNAL RADIUS
RE=0.2               ! EXTERNAL RADIUS
DX=0.13              ! RADIAL DISPLACEMENT

TREF=293             ! REFERENCE TEMPERATURE, K

/UPF,usermat.F
/UPF,usermatth.F

/PREP7
ET,1,SOLID226,11          ! STRUCTRAL-THERMAL COUPLING
KEYOT,1,9,1               ! SUPPRESSED THERMOELASTIC DAMPING

MP,ALPX,1,23.8E-6         ! THERMAL EXPANSION COEFFICIENT
MP,DENS,1,2700            ! DENSITY
MP,QRATE,1,0.9            ! TAYLOR-QUINNEY FACTOR

TB,USER,1,6,4             ! USER-DEFINED STRUCTURAL MATERIAL
TBTEMP,293
TBDATA,1,7.E10,0.3,70E6,210E6     ! E,NU,SIGY,ET
TBTEMP,313
TBDATA,1,7.E10,0.3,69.58E6,210E6  ! E,NU,SIGY,ET
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
TBTEMP,333
TBDATA,1,7.E10,0.3,69.16E6,210E6  ! E,NU,SIGY,ET
TBTEMP,353
TBDATA,1,7.E10,0.3,68.74E6,210E6  ! E,NU,SIGY,ET
TBTEMP,373
TBDATA,1,7.E10,0.3,68.32E6,210E6  ! E,NU,SIGY,ET
TBTEMP,393
TBDATA,1,7.E10,0.3,67.9E6,210E6   ! E,NU,SIGY,ET

TB,USER,1,1,2,THERM      ! USER-DEFINED THERMAL MATERIAL
TBDATA,1,150,900         ! K,C

TB,STATE,1,,8            ! EPEQ,EPPL,SEQV

RECT,RI,RE,0,RE-RI
TYPE,1
MAT,1
LESIZE,1,,,10
LESIZE,2,,,1
LESIZE,3,,,10
LESIZE,4,,,1
K,100,0
K,101,0,1
VROT,1,,, ,,,100,101,10,1
VMESH,1
ALLSEL,ALL

CSYS,5
NROTATE,ALL
NSEL,S,LOC,z,0
NSEL,A,LOC,z,RE-RI
D,ALL,Uz,0              ! CONSTRAIN UY DOF AT TOP AND BOTTOM FACES TO
                        ! SIMULATE INFINITELY LONG CYLINDER
NSEL,S,LOC,X,RI
D,ALL,UX,DX  ! APPLY RADIAL DISPLACEMENT ALONG INNER SURFACE OF CYLINDER

NSEL,S,LOC,Y,0
NSEL,A,LOC,Y,10
D,ALL,UY
ALLSEL,ALL
CSYS,0

TREF,TREF   ! REFERENCE TEMPERATURE
IC,ALL,TEMP,TREF       ! INITIAL CONDITION
FINISH

/COM,
/COM, ***** SOLUTION TIME = 13 SEC (ADIABATIC PROCESS) *****
/COM,
/SOLUTION
ANTYPE,TRANSIENT       ! TRANSIENT ANALYSIS
NLGEOM,ON              ! LARGE DEFLECTION ON
TINTP,,,,1.0           ! TRANSIENT INTEGRATION PARAMETER
KBC,0                  ! RAMPED LOADING
NSUB,20,100,5          ! NUMBER OF SUBSTEPS
OUTRES,ALL,ALL
CNVTOL,HEAT,1.E-2,1.E-2 ! CONVERGENCE TOLERANCE
TIME,13     ! END TIME
SOLVE
FINISH

/POST26
NSEL,S,LOC,X,0.1
NSEL,R,LOC,Y,0.1
ND=NDNEXT(0)
NSOL,2,ND,TEMP          ! TEMPERATURE AT NODE =ND
FILLDATA,3,,,,-1
FILLDATA,4,,,,293.0
PROD,5,3,4
ADD,6,2,5
NSOL,7,ND,U,X,UX        ! UX AT NODE=ND
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
PRVAR,7,6
/GRID,1
/AXLAB,X,Displacement of inner surface[m]
/AXLAB,Y,Temperature variation [K]
/XRANGE,0,0.14
/GROPT,DIVX,14
/YRANGE,0,60
/GROPT,DIVY,6
XVAR,7
PLVAR,6
ALLSEL,ALL
FINISH

/POST1
SET,LAST,LAST
PLNSOL,TEMP
PLNSOL,EPPL,EQV
FINISH
```

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Appendix E. Fully Coupled Wind Turbine Example in Mechanical APDL

This wind coupling example solution has been implemented in Mechanical APDL to illustrate how to perform an integrated analysis of a wind turbine with its supporting structure. In this solution procedure, both the structural code (that is, Mechanical APDL) and aeroelastic (e.g. Flex5) code are run simultaneously with continuous data exchange between the two programs at each time step. The data transfer is done through a set of interface routines that put or get data from a shared common data space. The interface is supplied in the form of a DLL which both the structural and aeroelastic programs will be accessing during an analysis. The example includes user programmable functions, and a macro has been developed to facilitate the communication with the DLL from Mechanical APDL. Aeroelastic code developers will need to utilize the same set of routines to establish communications between their code and the DLL.

There is also a sequential coupling method available; see Sequential Coupled Wind Turbine Solution in Mechanical APDL in the *Advanced Analysis Guide* for more information about using that method.

E.1. Implementing a Fully Coupled Wind Turbine Analysis

This example implementation of the fully coupled wind solution in Mechanical APDL follows a similar strategy to that used for ASAS. In particular, data access is provided through the same set of interface routines. This enables easy adaptation of the new facility by existing ASAS wind turbine users.

The following summarize the modeling characteristics in Mechanical APDL for a wind coupling analysis:

- The turbine effect is modeled via the user element USER300. This special user element has 9 nodes, with 6 freedoms (UX, UY, UZ, ROTX, ROTY, and ROTZ) on the first node and 3 freedoms (UX, UY, and UZ) on each of the subsequent nodes, making it capable of having a maximum 30 degrees of freedom on the element. The first node is the connection point between the turbine and the supporting structure and therefore it must be a node in the structural model. The other 8 nodes are created to accommodate the additional freedoms that are internal to the turbine element and are used solely by the aeroelastic code. Therefore, these nodes should not be connected to any other parts of the model.

- Key option 1 (KEYOPT(1)) of the user element is used to specify the damping matrix option. Damping can be obtained from the aeroelastic code alone, or computed from Rayleigh damping in Mechanical APDL based on the turbine mass and stiffness matrices, or both. KEYOPT(1) = 0 indicates that the damping matrix will be taken from the aeroelastic code plus Rayleigh damping, and this is the default. KEYOPT(1) = 1 indicates that only Rayleigh damping will be used. KEYOPT(1) = 2 indicates that only damping from the aeroelastic code will be used.

- The element does not have any material property or real constant data.

- The element mass will not generate any body forces even if accelerations (e.g. ACEL) are defined.

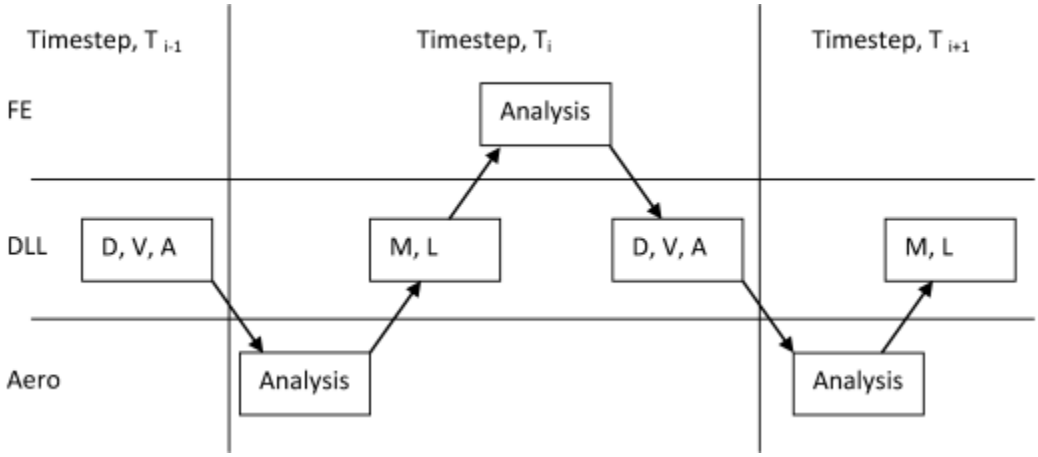
Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The only element results available are the element nodal forces.
- A Mechanical APDL command macro called **WTBCREATE** is provided to assist with the creation of a wind turbine model. This will automatically generate a turbine element and issue relevant data commands that are necessary to run a wind coupling analysis.
- Special versions of the user subroutines `UserElem`, `USolBeg`, `USsFin` and `USolFin` are provided to enable a wind coupling analysis. In addition, the shared common DLL `WTBFunctions.dll` is also required. A custom build of Mechanical APDL is required during which the aeroelastic linking option should be selected.
- The analysis type should be transient (**ANTYPE**, TRANS) for a wind coupling analysis.

E.2. Theory

The whole offshore wind turbine structure is split into two parts. The upper part is the rotor-nacelle-assembly of the turbine including the tower and this is modeled by a wind turbine aeroelastic code.[1] The lower part is the support structure and this is modeled by structural FE as usual.

In this coupling approach, both the aeroelastic and FE code runs concurrently, with data exchange between the two programs occurring at every time step. The turbine effect as modeled by the aeroelastic code is taken into account in the FE code as a special element, which accepts the wind turbine system matrices (stiffness, damping and mass) and aerodynamic load vector as if it is a superelement.



In the diagram above, the following data items are exchanged:

D, V, A: Displacements, Velocities, Acceleration information at interface point.

M, L: Matricies (Stiffness, Damping and Mass), and/or Load Vectors.

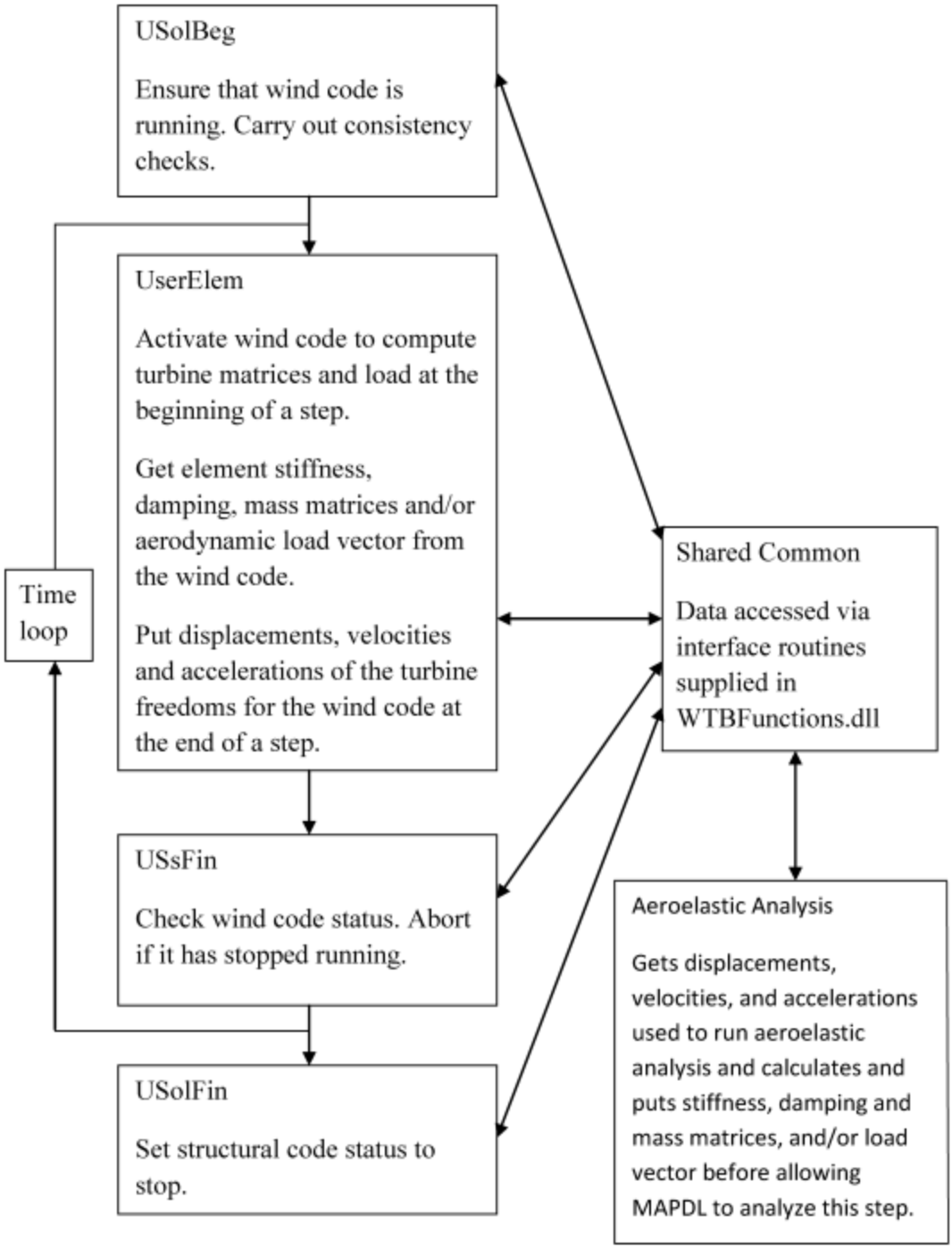
At the beginning of a time step, the aeroelastic code is called to compute the turbine matrices and loading based on the kinematics at the end of the previous step. It is assumed that the turbine data will remain constant during the time step. After transferring the information to the FE code, the full system of equations of motion is solved by the FE code to obtain an updated solution. Finally, the turbine kinematics are passed back to the aeroelastic code to continue the time advancement process.

[1] In the case of Flex5, the maximum number of equations generated by a model is 28.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Further details about the theory of the fully coupled solution can be obtained from the work of Kaufer [1] and Seidal [2].

Flowchart of Wind Coupling Calculations



E.3. Compiling a Custom Version of Mechanical APDL

To activate the wind coupled analysis feature, it is necessary to create a custom build of Mechanical APDL that includes the wind interface libraries `Aeroelastic.lib` and `WTBFunctions.lib`. The

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

library `Aeroelastic.lib` contains updated user routines to enable a wind coupling analysis in Mechanical APDL. The library `WTBFunctions.lib` contains interface routines for storing and retrieving data to a shared common area. Since the wind coupling solution involves changes to some of the user routines, it is important that the library `Aeroelastic.lib` is linked in before all other user libraries so that the wind library versions are picked up by the linker.

The executable of the aeroelastic code should be placed in the same folder as the custom Mechanical APDL executable as both programs need to use `WTBFunctions.dll` to access the shared data. Alternatively, the folder containing the WTBFunctions.dll could be added to the path (on Windows systems) to enable the aeroelastic code to remain elsewhere.

To link a custom version, run ANS_ADMIN from the utilities folder in the start menu. During the relinking, the user is asked if the aeroelastic functions are to be included.

For further information and compiler requirements regarding the linking procedure and additional settings for using custom compiled executables, see Compiling and Linking UPFs on Windows Systems in the *Programmer's Reference* (p. 139).

E.4. Performing a Wind Coupled Analysis

In order to perform a wind coupling analysis, the aeroelastic software must be modified to provide data communication with Mechanical APDL through the specified interface.

The aeroelastic analysis should be started before the FE analysis Both programs should use the same `WTBFunctions.dll` at run time.

E.4.1. The Wind Coupling Process

Below is a brief description of the wind coupling algorithm implemented in Mechanical APDL:

1. Both Mechanical APDL and the aeroelastic code are run concurrently.
2. Both programs carry out data initialization separately.
3. At the beginning of a time step, set the aeroelastic code to active. The aeroelastic code computes and returns the turbine stiffness, damping and mass matrices to Mechanical APDL together with the aerodynamic force vector for the current time step. Put the aeroelastic code to sleep.
4. Mechanical APDL carries out the time integration to find the new solution at the end of the current time step.
5. The displacements, velocities and accelerations of the turbine freedoms (i.e. freedoms on the USER300 element) at the end of the step are stored to the shared common for the aeroelastic code.
6. Advance to next time step and return to step 3.

E.4.2. Data Exchange Routines

The following routines are used to facilitate data exchange between Mechanical APDL and the aeroelastic code as described above. They can be accessed from the aeroelastic code by linking in the

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

dynamic link library `WTBFunctions.dll`. Coding examples utilizing these routines in FORTRAN and C++ are available in the folder `Program Files\Ansys Inc\v212\ansys\custom\user\{platform}\Aeroelastic`, where {platform} is a directory that uniquely identifies the hardware platform version: "Winx64" for 64-bit Windows.

```
SUBROUTINE GetWTBParamI(itype,id,ival,ierr)
!****
!****    Routine gets an integer wind turbine parameter to common data area
!****
!****    Arguments
!****    itype  (in ) Data type
!****                 1 - Mechanical APDL run status
!****                     = -1 not started
!****                     =  0 inactive (waiting)
!****                     =  1 active (running)
!****                     =  2 stopped (finished/aborted)
!****                 2 - Wind code run status
!****                     = -1 not started
!****                     =  0 inactive (waiting)
!****                     =  1 active (running)
!****                     =  2 stopped (finished/aborted)
!****                 3 - Number of active freedoms
!****                 4 - Time step number
!****    id     (in ) element identifier (currently unused, assume 1)
!****    ival   (out) parameter value
!****    ierr   (out) exit code (0 if no error)
integer itype,id,ival,ierr


SUBROUTINE GetWTBParamR(itype,id,rval,ierr)
!****
!****    Routine gets a real wind turbine parameter to common data area
!****
!****    Arguments
!****    itype  (in ) Data type
!****                 1 - Analysis time
!****                 2 - Time step
!****    id     (in ) element identifier (currently unused, assume 1)
!****    rval   (out) parameter value
!****    ierr   (out) exit code (0 if no error)
integer itype,id,ierr
double precision rval


SUBROUTINE GetWTBData(itype,id,array,narray,ierr)
!****
!****    Routine gets wind turbine data from common data area
!****
!****    Arguments
!****    itype  (in ) Data type
!****                 1 - Stif
!****                 2 - Damp
!****                 3 - Mass
!****                 4 - Load
!****                 5 - Disp
!****                 6 - Velo
!****                 7 - Accn
!****    id     (in ) element identifier (currently unused, assume 1)
!****    array  (out) data array
!****    narray (i/o) size of array on input, actual array size on exit
!****    ierr   (out) exit code (0 if no error)
integer itype,id,narray,ierr
double precision array(*)


SUBROUTINE PutWTBParamI(itype,id,ival,ierr)
!****
!****    Routine puts an integer wind turbine parameter to common data area
!****
!****    Arguments
```

```
!****    itype  (in ) Data type
!****                   1 - Mechanical APDL run status
!****                       = -1 not started
!****                       =  0 inactive (waiting)
!****                       =  1 active (running)
!****                       =  2 stopped (finished/aborted)
!****                   2 - Wind code run status
!****                       = -1 not started
!****                       =  0 inactive (waiting)
!****                       =  1 active (running)
!****                       =  2 stopped (finished/aborted)
!****                   3 - Number of active freedoms
!****                   4 - Time step number
!****    id     (in ) element identifier (currently unused, assume 1)
!****    ival   (in ) parameter value
!****    ierr   (out) exit code (0 if no error)
integer itype,id,ival,ierr


SUBROUTINE PutWTBParamR(itype,id,rval,ierr)
!****
!****    Routine puts a real wind turbine parameter to common data area
!****
!****    Arguments
!****    itype  (in ) Data type
!****                   1 - Analysis time
!****                   2 - Time step
!****    id     (in ) element identifier (currently unused, assume 1)
!****    rval   (in ) parameter value
!****    ierr   (out) exit code (0 if no error)
integer itype,id,ierr
double precision rval


SUBROUTINE PutWTBData(itype,id,array,narray,ierr)
!****
!****    Routine puts wind turbine data to common data area
!****
!****    Arguments
!****    itype  (in ) Data type
!****                   1 - Stif
!****                   2 - Damp
!****                   3 - Mass
!****                   4 - Load
!****                   5 - Disp
!****                   6 - Velo
!****                   7 - Accn
!****    id     (in ) element identifier (currently unused, assume 1)
!****    array  (in ) data array
!****    narray (i/o) size of array on input, actual put size on exit
!****    ierr   (out) exit code (0 if no error)
integer itype,id,narray,ierr
double precision array(*)
```

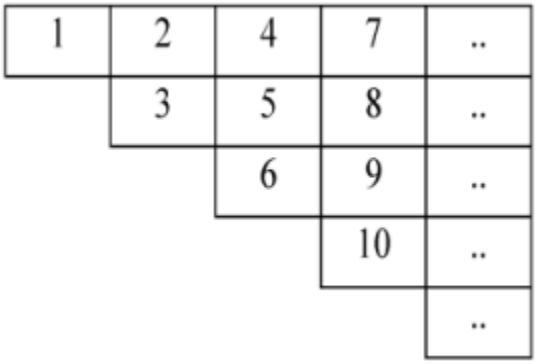
E.4.3. Important Analysis Notes

You must keep in mind the following when performing an aeroelastic analysis:

- After data initialization (e.g. data read in and checking, etc), the aeroelastic code should be put to sleep until the Mechanical APDL run status becomes inactive.
- The number of active freedoms is the number of freedoms in the aeroelastic model. This must be set up and put to the shared common by the aeroelastic code during the data initialization phase of the analysis.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- For the current usage, the aeroelastic code should always only put stiffness, damping, mass, and load data to the shared common, and get displacements, velocities, and accelerations from the shared common.
- The wind turbine array entries must correspond to the order of the element freedoms set up for the wind coupled USER300 element. Thus, freedoms 1 to 6 are UX, UY, UZ, ROTX, ROTY, and ROTZ freedoms of the interface node between the turbine and the support structure. The rest are generalized freedoms internal to the element. All the data must be stated in the structural coordinate axis system.
- The element matrices (i.e. stiffness, damping, and mass) are assumed to be given in packed symmetric form. The order of the packed symmetric matrix form in which the data are specified is defined as follows:



- The units of the wind turbine data values are assumed to be consistent with the analysis units. No units conversion will be carried out by Mechanical APDL.
- It is assumed that identical time step sizes are used in both Mechanical APDL and the aeroelastic code. The solution times are controlled by data in Mechanical APDL since it is the one that solves the complete set of equations of the coupled system.
- The table below shows the explanation of the various exit code values (i.e. ierr):

Code	Meaning
100	Invalid data type integer
101	Specified array size too small to get
102	Specified array size too big to put
103	Number of active freedoms is unset
104	Invalid array size specified
201	Invalid run status specified
202	Invalid number of active freedoms specified
203	Invalid time step number specified
301	Invalid time step value specified

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

E.5. Example Analysis Using Provided "WindProg" Example for Aeroelastic Coupling

This example uses `WindProg.exe`, which represents a dummy aeroelastic analysis. Source code to allow you to compile this program is provided for both C++ and FORTRAN in `{Installation Folder}\ansys\custom\user\{Platform}\Aeroelastic\WindDemos\{C++|FORTRAN}`. It assumes that the steps in Compiling a Custom Version of Mechanical APDL (p. 411) have been followed to create a custom executable for Mechanical APDL.

The example Mechanical APDL file is a simple line of pipe elements which is fully supported at node 6 and has the interface node positioned at node 1. The **WTBCREATE** macro is used to set up the user element and define the interface node.

To run the analysis, `WindProg.exe` must be able to find the `WTBFunctions.dll` either by having both files in the same folder, or by adding the folder containing `WTBFunctions.dll` to your Path system environment variable. First run WindProg and enter 3 as the number of freedoms, then run the following example within Mechanical APDL.

```
/FILNAME,wind02
/prep7
/TITLE,wind02, Wind coupling test
/com ************************************************************************
/com Wind coupling testing 02
/com ************************************************************************
antype,trans
nlgeom,off
et,1,pipe288
! define pipe section
secnum,1
sectype,1,pipe
secdata,0.1,0.02
MP,EX,  1,2.1e11
MP,PRXY,1,0.3
MP,ALPX,1,0.0
MP,DENS,1,7850.0
! define the tube
n,        1,        0.0,   0.0
n,        2,        1.0,   0.0
n,        3,        2.0,   0.0
n,        4,        3.0,   0.0
n,        5,        4.0,   0.0
n,        6,        5.0,   0.0
type,1
mat,1
secnum,1
en, 1,   1,  2
en, 2,   2,  3
en, 3,   3,  4
en, 4,   4,  5
en, 5,   5,  6
! define damping factors
alphad,0.3
betad,0.001
! define turbine element
wtbcreate,,1,1  ! use Rayleigh damping
d,  6,all
finish
/SOLU
! CASE 1
F,1,FY,1.0e5
TINTP,0.0
TIME,1.0e-6
nsubst,1
solve
```


Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
TIME,0.1
nsubst,10,10,10
OUTPR,all,1
OUTRES,all,1
solve
finish
/POST1
FORCE,STATIC
PRESOL,F
PRESOL,M
finish
```

E.6. References

The following references are cited in this appendix:

1. D. Kaufer et. al., Integrated Analysis of the Dynamics of Offshore Wind Trubines with Arbitrary Support Structures, Proc. of EWEC 2009, Marseille: EWEC, 2009.

2. M. Seidal et. Al., Validation of Offshore Load Simulations Using Measurement Data from the DOWNVInD Project, Proc. European Offshore Wind 2009, Stockholm, 2009.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2021 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.