Collaborative
Solutions Inc

An ANSYS Support Distributor

Date December 26, 1999 Memo Number STI23:991226B
Subject **ANSYS Tips & Tricks: Mass Calculation Macros**
Keywords General: xSUM: PSOLVE: Mass Properties

1. Introduction:

During an analysis, verification of mass properties in the model is a simple way to ensure that material properties and geometries have been input/created properly. This memo covers some ways to determine mass properties of an ANSYS model.

2. Summary:

There are three methods covered in this memo by which users can determine mass property information – the use of the xSUM family of commands to obtain information from the solid model or by performing partial/full solutions to determine this information from the finite element mesh during element matrix formation. There are different limitations among each of the methods (speed vs. accuracy considerations), so the user may need to find the most appropriate method for his/her uses. The reader is referred to Ch. 15.15 "Mass Moments of Inertia" of the Theory Manual as well as the Commands Reference Manual for more information on this topic.

3. General Discussion:

Three methods in which a user can calculate mass properties of a model are (1) using the xSUM commands, (2) performing a partial/full solution and (3) using the IRLF command and performing a partial/full solution.

3.1 Discussion of xSUM Commands:

There are a series of preprocessing commands in ANSYS which allow the user to determine mass properties of solid model geometry: KSUM, LSUM, ASUM, and VSUM for keypoint, line, area, and volume calculations, respectively. With appropriate element attributes set, the ASUM and VSUM commands can provide useful information about geometry statistics.

Generally speaking, the KSUM and LSUM commands are not as useful as ASUM and VSUM. Only the latter two incorporate element attributes in its calculations. If a user needs mass properties of beam elements, please see the other two methods discussed below.

While in the Preprocessor, the user can perform the following:

1) Assign element attributes to the areas/volumes (this includes density and, for shells, thickness)
2) Select areas/volumes which will be included in the mass calculations
3) Issue the ASUM/VSUM commands to calculate mass properties for the selected entities

If no density or shell thickness is assigned, ANSYS assumes a unit density and/or thickness by default. The degree of tessellation is determined by the /FACET command by default, but a user can use the LAB=FINE argument to always produce more accurate calculations irrespective of the /FACET setting. Note that using ASUM,FINE or VSUM,FINE will result in longer calculation times. These commands can be used prior to or after meshing since the calculation accounts for solid model geometry only (not the finite element mesh).

There are a few limitations of the xSUM family of commands, all of which the user should keep in mind. As mentioned above, since these commands calculate geometry statistics of the selected solid model entities, not all element types are supported. These include mass, beam, and composite shell/solid elements (MASS21, BEAMx, SHELL91/99, SOLID46). To account for calculations for these elements, one of the two methods discussed later should be pursued.

Also, because the xSUM commands focus on the solid model geometry, the calculations may not be exactly representative of the underlying finite element mesh. This is due to the fact that, for a coarse mesh, the discretization of the solid model may be approximate. The xSUM commands rely solely on the solid model geometry and can be used before or after meshing since the calculations are irrespective of the mesh. As a result, the user should keep in mind that for coarser meshes, the calculations from the xSUM commands may be slightly off from the actual finite element mesh.

An example output from VSUM,FINE for a specific model is printed below:

```
PRINT GEOMETRY ITEMS ASSOCIATED WITH THE CURRENTLY SELECTED VOLUMES
   DENSITY FOR VOLUME      1  IS  0.25900E-03

   TOTAL NUMBER OF VOLUMES SELECTED =     1  (OUT OF      1 DEFINED)

   TOTAL VOLUME OF ALL SELECTED VOLUMES =   2000.0
   TOTAL MASS =  0.51800

   CENTROID:  XC=  10.000      YC=  5.0000      ZC=  5.0000

                    *** MOMENTS OF INERTIA ***

            ABOUT ORIGIN     ABOUT CENTROID       PRINCIPAL
     IXX =    34.533           8.6333             8.6333
     IYY =    86.333           21.583             21.583
     IZZ =    86.333           21.583             21.583
     IXY =   -25.900         -0.26041E-11
     IYZ =   -12.950         -0.13021E-11
     IZX =   -25.900         -0.26006E-11

     PRINCIPAL ORIENTATION VECTORS (X,Y,Z):
       1.000  0.000  0.000    0.000  1.000  0.000    0.000  0.000  1.000
       (THXY=    0.000  THYZ=    0.000  THZX=    0.000)
```

3.2 Discussion of Standard Solution Output:

Upon initialization of the solution, mass property information is usually printed out in the output window (or output file). This provides an approximate summary of masses and moments of inertia.

These calculations are approximate due to the fact that some accuracy is compensated for speed. Moreover, varying mass in different directions and rotational inertia terms (both of which can occur with MASS21 elements) are not included (note that only one number is listed for "total mass"). Also, centroid calculations on a per-element basis are determined by the average of attached nodes, so they are not precise for trapezoidal elements. Lastly, centroid calculations do not include the effects of beams and shells with offsets (i.e., effects from offsets are not included). While most of these limitations are not critical, it is important for the user to keep these in mind.

If a user does not want to perform an actual solution but wants to calculate mass properties only, the following macro can be used:

```
/solu
/output,mass_output,txt
psolve,elform
/output
finish
```

The /OUTPUT commands are optional but are used to save/redirect the information to a text file called "mass_output.txt" for future reference. This is important for Windows platforms since the DOS command prompt does not have the ability to redirect output both to screen and file as with the xterm utility on UNIX machines, so a user may want to save output to a file if it is needed in the future. The PSOLVE command in the above macro is used to perform a partial solution (instead of a full solution) to create the element matrices only. From this calculation, mass summaries can be obtained. This is a less expensive way to obtain the information instead of performing an actual solution.

An example output from a solution for the same model as Section 3.1 is printed below:

```
***** CENTROID, MASS, AND MASS MOMENTS OF INERTIA *****

  CALCULATIONS ASSUME ELEMENT MASS AT ELEMENT CENTROID

  TOTAL MASS =  0.51800

                           MOM. OF INERTIA         MOM. OF INERTIA
```

```
   CENTROID              ABOUT ORIGIN           ABOUT CENTROID

XC =   10.000          IXX =    34.45          IXX =    8.547
YC =   5.0000          IYY =    86.25          IYY =    21.50
ZC =   5.0000          IZZ =    86.25          IZZ =    21.50
                       IXY =   -25.90          IXY =  -0.3020E-12
                       IYZ =   -12.95          IYZ =  -0.3357E-12
                       IZX =   -25.90          IZX =  -0.2984E-12


*** MASS SUMMARY BY ELEMENT TYPE ***

TYPE      MASS
   1  0.518001
```

From the output, note that the moment of inertia calculations differ slightly than in Section 3.1 above. This illustrates the trade-off between accuracy and speed for this given mesh.

3.3 Discussion of Inertial Relief Solution Output:

Similar to the method discussed in Section 3.2 above, another, more accurate, calculation of masses and moments of inertia can be performed upon solution as the element matrices are formed. This method utilizes the IRLF,-1 command. While the IRLF command is generally used for inertia relief, this command can be used to obtain more accurate mass property information than the standard solution method.

Prior to solving a model (or performing a partial solve), the IRLF,-1 command should be issued. This tells ANSYS to use the mass matrices directly in the calculations. This includes the effect of mass in different directions and rotational inertial terms (see output below; mass is accounted for in x-, y-, and z-directions). However, as with the 'regular' method outlined above, the effect of offsets in centroid calculations for beams and shells are not included.

For users who wish to perform this calculation independent of an actual solution, the following macro can be used:

```
/solu
outpr,basic,all
irlf,-1
/output,mass_output,txt
psolve,elform
psolve,elprep
/output
finish
```

The OUTPR command ensures that basic information is printed out (see note below). The IRLF,-1 command tells ANSYS not to perform actual inertia relief calculations but precise mass and moment of inertia calculations only. The /OUTPUT command redirects information to a text file for later use (see note between Windows and UNIX in Section 3.2 above for more details). An alternative to redirecting output is using the IRLIST command to list the mass calculation results after the partial/full solution has been completed. While the "regular" method to calculate approximate mass and centroid information involves only the element formation process (PSOLVE,ELFORM), the "accurate" method requires an additional partial solution command (PSOLVE,ELPREP) for its calculations (as specified by IRLF,-1).

An example output from a solution for the same model as Section 3.1 is printed below:

```
***********  PRECISE MASS SUMMARY  ***********

 TOTAL MASS (X,Y,Z)= 0.51800     0.51800     0.51800

 MOMENTS AND PRODUCTS OF INERTIA TENSOR (I) ABOUT ORIGIN (GLOBAL CARTESIAN)
      34.533     -25.900     -25.900
     -25.900      86.333     -12.950
     -25.900     -12.950      86.333
```

```
CENTER OF MASS (X,Y,Z)=  10.000      5.0000      5.0000

MOMENTS AND PRODUCTS OF INERTIA TENSOR (I) ABOUT CENTER OF MASS (GLOBAL CARTESIAN)
     8.6333     0.28777E-11 0.29772E-11
    0.28777E-11  21.583     0.15259E-11
    0.29772E-11 0.15259E-11  21.583

PRINCIPAL CENTROIDAL MOMENTS OF INERTIA=  8.6333      21.583      21.583

ORIENTATION VECTORS (GLOBAL CARTESIAN) FOR THE PRINCIPAL AXES=
  ( 1.000, 0.000, 0.000) ( 0.000, 1.000, 0.000) ( 0.000, 0.000, 1.000)

ANGLES (XY,YZ,XZ) OF THE PRINCIPAL AXES=  0.000      0.000      0.000


         *********  TOTAL LOAD SUMMARY  *********
                                          X-AXIS      Y-AXIS      Z-AXIS
FORCES AT CENTER OF MASS.................. 0.45963E-12 -22000.    -0.38177E-11
MOMENTS ABOUT ORIGIN...................... 0.11000E+06 0.51842E-10-0.28667E+06
MOMENTS ABOUT CENTER OF MASS.............-0.16221E-07 0.11367E-10 -66667.
```

Note that the centroid calculations match the calculations from the xSUM family of commands exactly. This is due to the fact that the geometry is very simple (blocks meshed with hexahedral elements). However, depending on the level of discretization, these results may differ from xSUM calculations for complex geometries with coarser meshes, as stated in Section 3.1 above. Also, the OUTPR,BASIC,ALL command will generate the "Total Load Summary", if desired. This summarizes force and moment information, similar to output obtained with the FSUM command in the General Postprocessor.

Also, upon executing this method, it will be noted that this method, though accurate, involves more calculations, so will be a little slower than the first method outlined in Section 3.2 above.

4. Conclusion:

Depending on the degree of accuracy or speed required, ANSYS allows the user to generate mass property information via three methods outlined in this memo – one procedure utilizes the solid model geometry information (Section 3.1) whereas the other two directly use the finite element mesh information (Sections 3.2 and 3.3). Macros can be used as noted in Sections 3.2 and 3.3 to facilitate this process if the user is only interested in mass and centroid information, rather than spending time in performing the entire solution.

Although not covered here, it is worth noting that there are other methods by which a user can access mass information. Alternative methods involve using APDL (ANSYS macro language) to *GET volume and density values per element in a *DO loop. Another technique entails setting up an element table of volume (ETABLE, , VOLU) for each set of material properties and performing the required operations (e.g., SMULT, SSUM). To keep this memo simple, only the more common techniques have been discussed.

5. References:

1) "ANSYS Theory Reference", Ch. 15 "Mass Moments of Inertia"
2) "ANSYS Commands Reference"

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

ANSYS Tips and Tricks

"ANSYS Tips & Tricks" is provided for customers of Collaborative Solutions, Inc. (CSI) with active TECS agreements, distributed weekly in Adobe Acrobat PDF format via email. Unless otherwise stated, information contained herein should be applicable to ANSYS 5.4 and above, although usage of the latest version (5.6 as of this writing) is assumed. Users who wish to subscribe/unsubscribe can send an email to operator@csi-ansys.com with their name and company information with the subject heading "ANSYS Tips and Tricks subscription". Older issues may be posted on CSI's website at http://www.csi-ansys.com.

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

XANSYS Mailing List

The xansys mailing list is a forum for questions and discussions of the use of ANSYS. As of 12/99, there are more than 900 subscribers with topics ranging from Structural, Thermal, Flotran, to Emag analyses, to name a few. Users are encouraged to subscribe to evaluate the usefulness of the mailing list for themselves. Also, either (a) using the mail program to filter [xansys] messages or (b) using the "digest" option to receive one combined email a day is strongly recommended to minimize sorting through the volume of postings.

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

To SUBSCRIBE: send blank email to xansys-subscribe@onelist.com
To unsubscribe send blank email to xansys-unsubscribe@onelist.com
Archived on http://www.infotech.tu-chemnitz.de/~messtech/ansys/ansys.html
ANOTHER archive on http://www.eScribe.com/software/xansys/
(A poor archive is also at http://www.onelist.com/archives.cgi/xansys)
```

CSI ANSYS Technical Support

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

```
Phone:  760-431-4815 (ask for ANSYS technical support)
Fax:  760-431-4824
Web:  http://www.csi-ansys.com
E-mail:  firstname.lastname@csi-ansys.com
Anonymous ftp site: ftp://ftp.csi-ansys.com

CSI Engineers:
Karen Dhuyvetter
Greg Miller
Sean Harvey
Alfred Saad
Bill Bulat
Sheldon Imaoka
David Haberman
```

All comments and suggestions are welcome.