Element Tables

Use of Element Tables in ANSYS

Sheldon Imaoka
Collaborative Solutions, Inc.
09/14/00

Definition of Element Tables

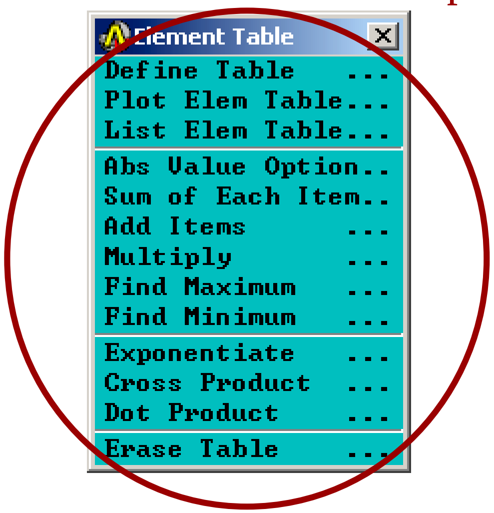
- An Element Table (ETABLE) is a “spreadsheet” of element information within ANSYS. These static arrays of information can be listed, printed, or operated upon for the following purposes:
  - ETABLE can be used to manipulate results for postprocessing, such as plotting “factors of safety” instead of stress values
  - Some elements have information which is pertinent only to that element. Hence, ETABLE is needed to retrieve these values. An example is obtaining moment and shear diagrams for BEAM elements.
  - Element tables are static tables, so they can be used to compare different results with each other (basic element table values can be updated automatically)

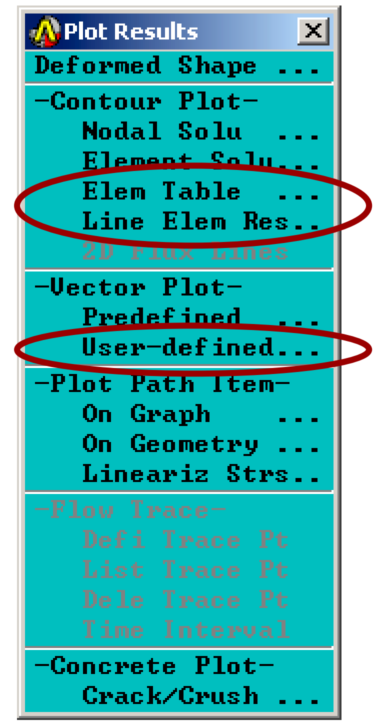
Usage of ETABLE

How to Access ETABLE

- ETABLE commands are accessible in the following locations:

“Main Menu > General Postproc > Element Table”





“Main Menu > General Postproc > Plot Results”

Steps Involved in Creating ETABLEs

1) Solve problem and load results set (as needed)

2) Define element table(s)

   – "Main Menu > General Postproc > Element Table > Define Table"

3) Perform operations on tables as required

   – "Main Menu > General Postproc > Element Table" allows summing, adding, multiplying, taking vector or cross product of tables, just to name a few examples.

4) Plot or list element tables

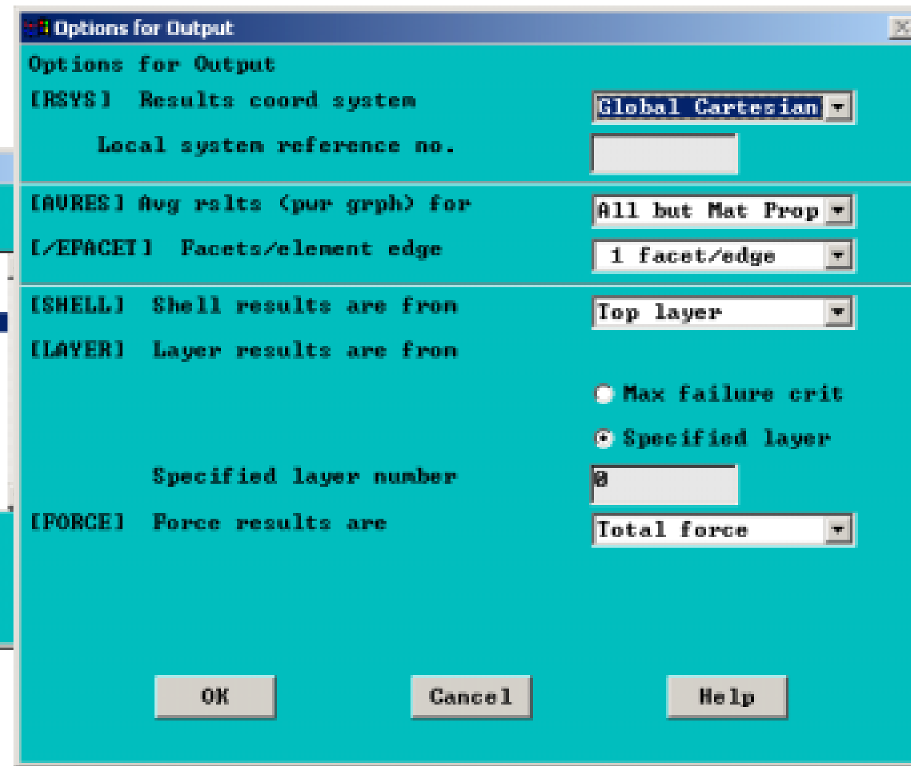
   – Plotting includes contour plots, line element results plots, or vector plots in "Main Menu > General Postproc > Plot Results"

Steps to Create ETABLE

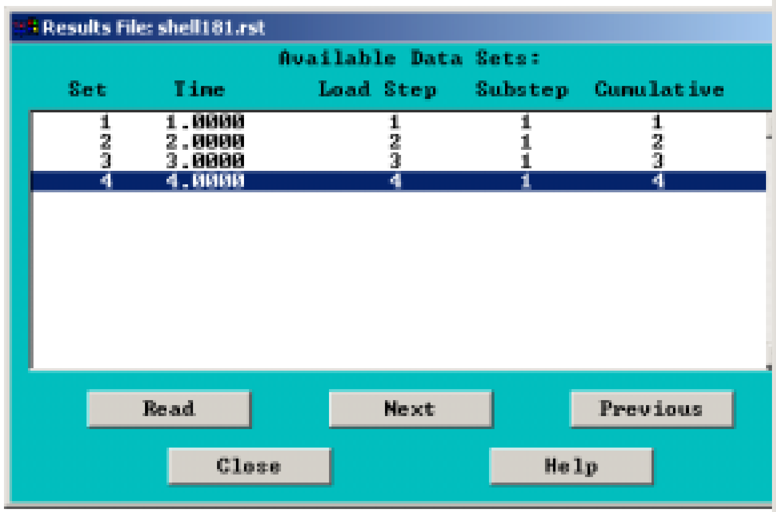
1) Solve problem and load results set (as needed)

– Define results coordinate system, layer for SHELL elements

General Postproc > Options for Outp…



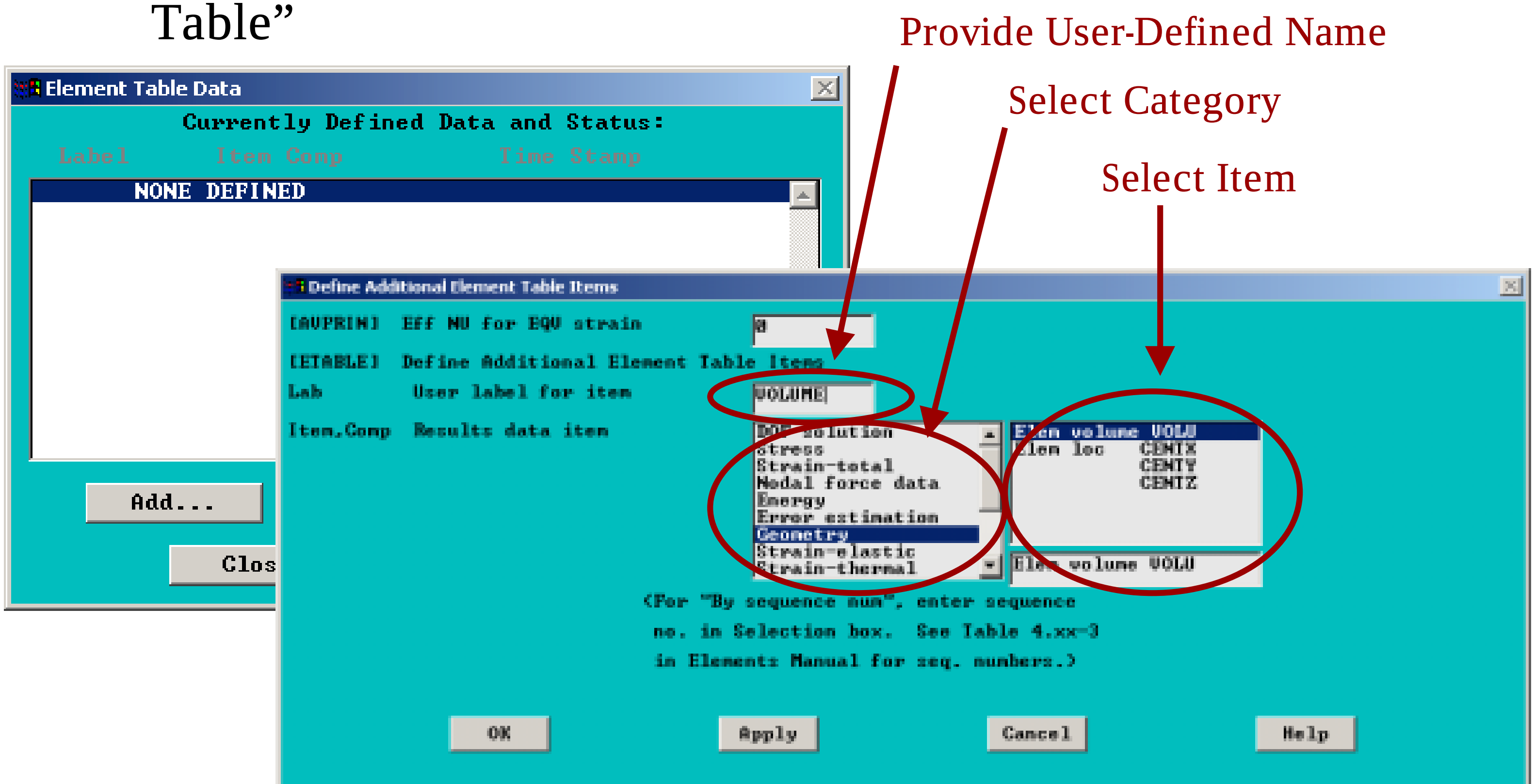
General Postproc > Results Summary…



Steps to Create ETABLE (cont.)

2) Define element table(s)

- “Main Menu > General Postproc > Element Table > Define Table”

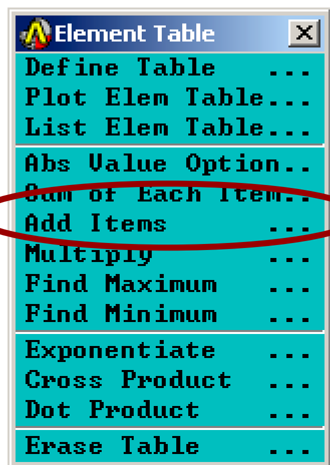


Steps to Create ETABLE (cont.)

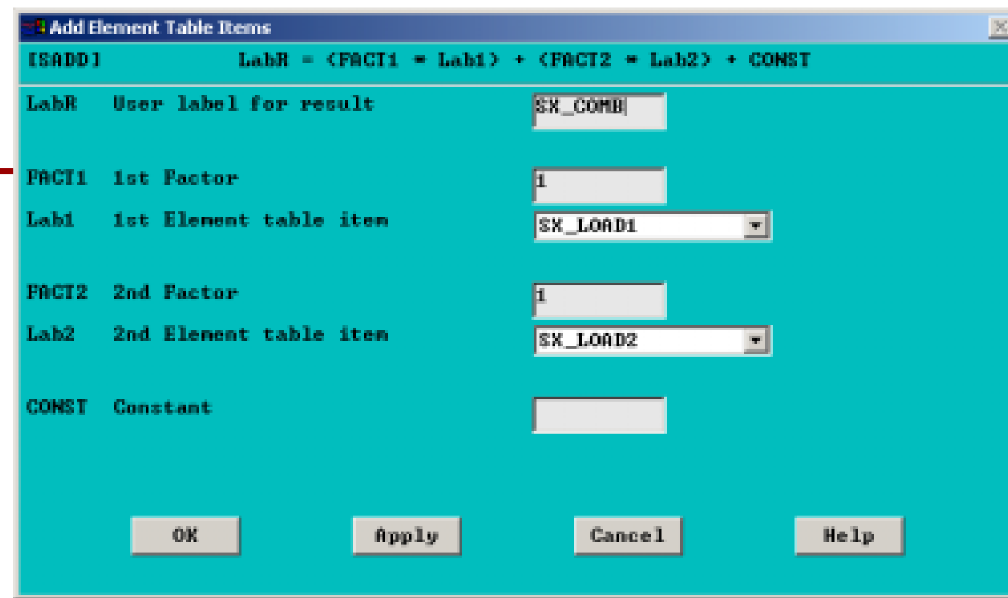
3) Perform operations on tables as required

– Sum, add, multiply, take vector or cross product of tables.

General Postproc > Element Table



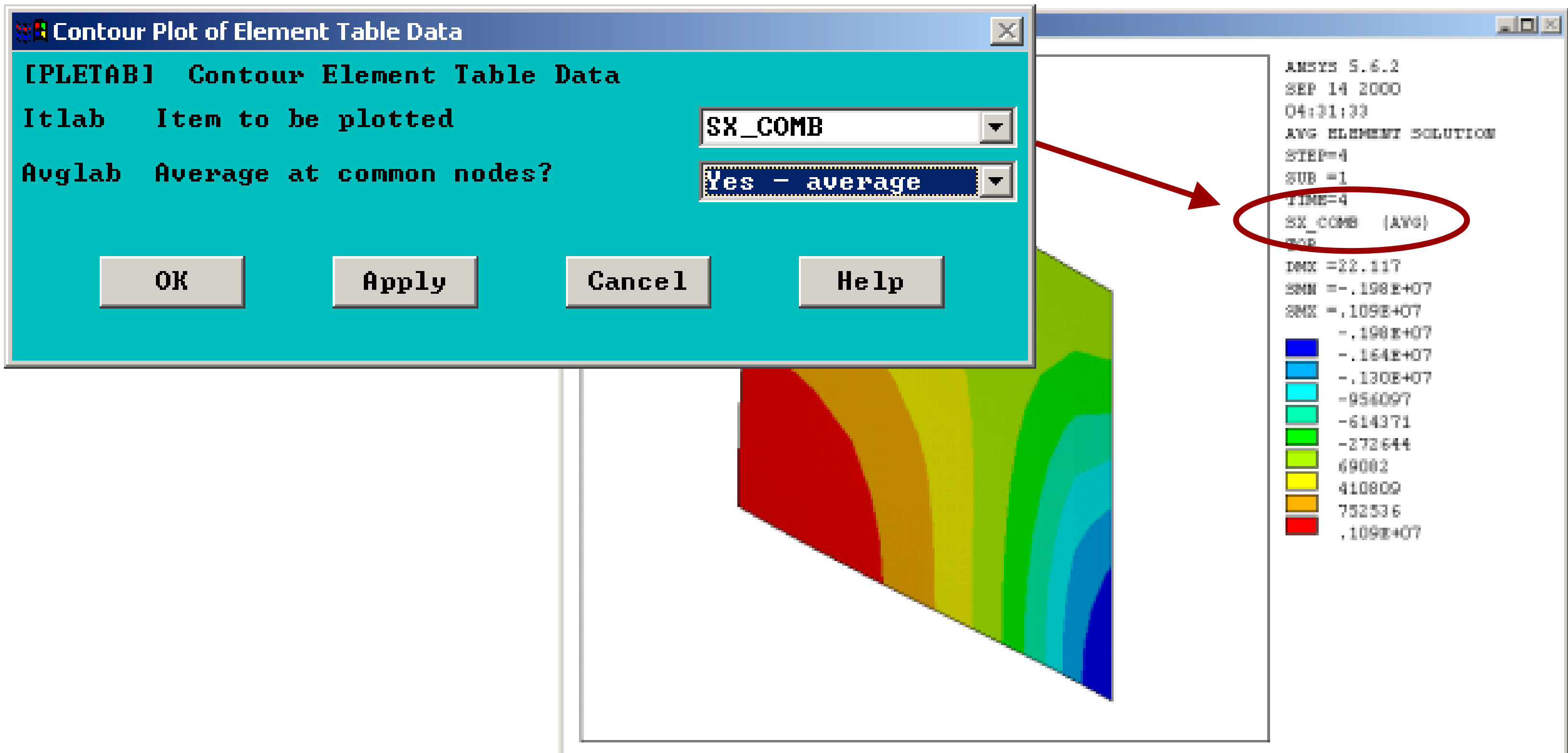
Example of Adding two element tables



Steps to Create ETABLE (cont.)

4) Plot or list element tables

- Plotting can be performed with or without averaging



Obtaining Element-Specific Data

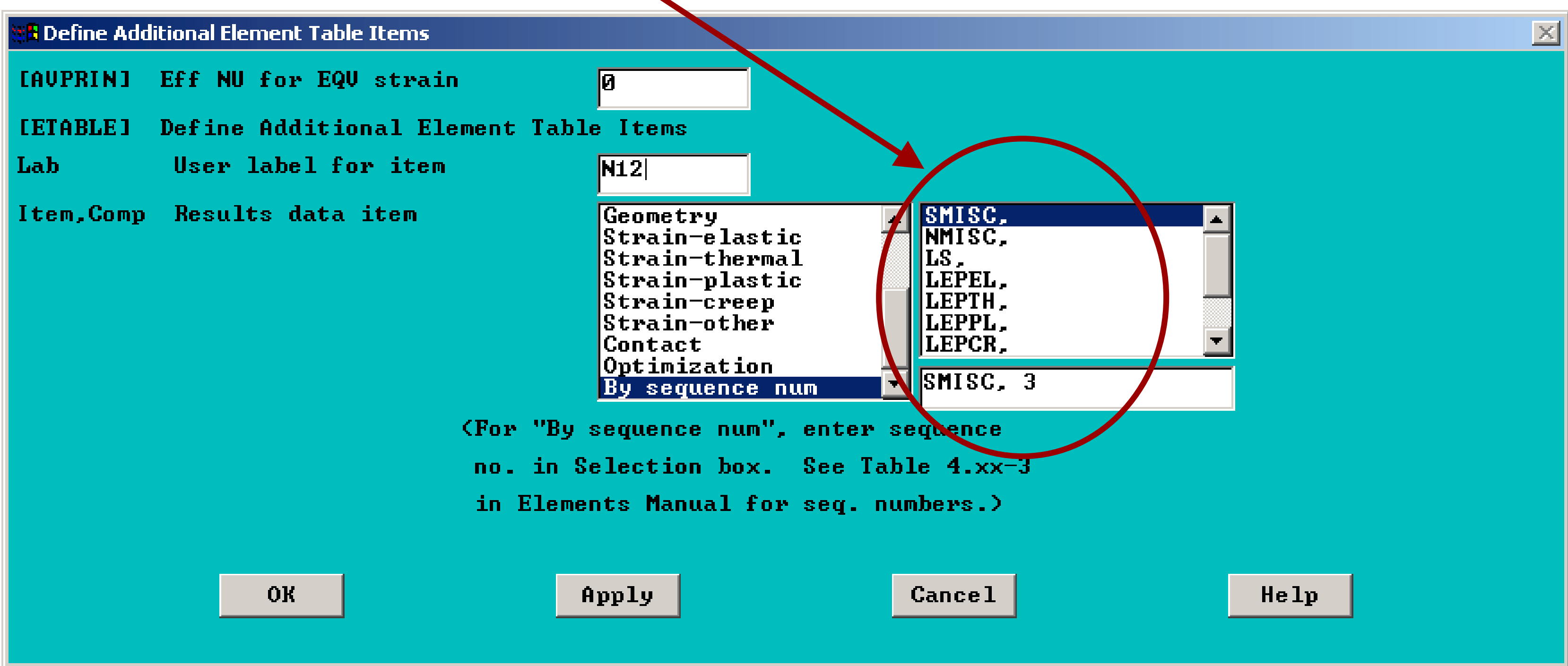
Table 3. SHELL181 Item and Sequence Numbers for the ETABLE and ESOL Commands

Name	Item	E	I	J	K	L
N11	SMISC	1	--	--	--	--
N22	SMISC	2	--	--	--	--
N12	SMISC	3	--	--	--	--
M11	SMISC	4	--	--	--	--
M22	SMISC	5	--	--	--	--
M12	SMISC	6	--	--	--	--
Q13[1]	SMISC	7	--	--	--	--
Q23[1]	SMISC	8	--	--	--	--
ε_{11}	SMISC	9	--	--	--	--
ε_{22}	SMISC	10	--	--	--	--
ε_{12}	SMISC	11	--	--	--	--
k_{11}	SMISC	12	--	--	--	--
k_{22}	SMISC	13	--	--	--	--
k_{12}	SMISC	14	--	--	--	--
γ_{13}	SMISC	15	--	--	--	--
γ_{23}	SMISC	16	--	--	--	--
THICK	SMISC	17	--	--	--	--
P1	SMISC		18	19	20	21

- Look up element information in online help.
- There will be a section for each element entitled “Output Data”
- A list of element-specific data (“Name”) will be shown with “Item” such as “SMISC” or “NMISC”

Obtaining Element-Specific Data (cont.)

- For this example, a user wants N12, defined as “in-plane element stress resultant” for SHELL181 in the online help. Use the information in the table “SMISC, 3” from the chart in the previous page to obtain “N12”.



Examples of ETABLE

BEAM189 Example

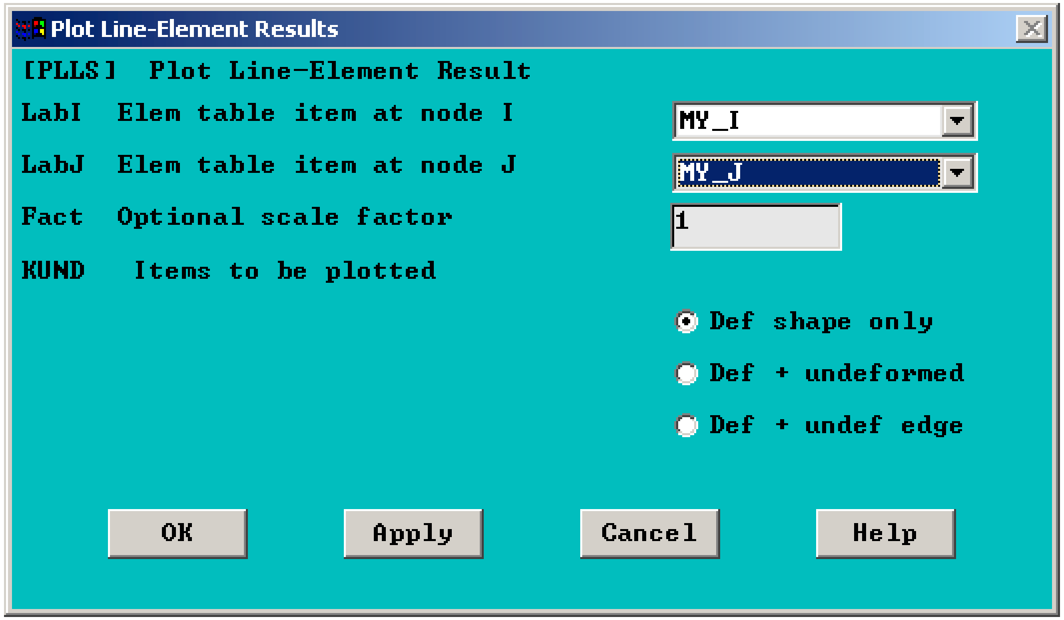
- BEAM elements can be plotted with bending moment or shear force diagrams. Bending for Node I and J are noted below as SMISC,2 and SMISC,15, respectively

Table 2. BEAM189 Item and Sequence Numbers for the ETABLE and ESOL Commands

Name	Item	I	J
Axial Force	SMISC	1	14
Bending Moment *My*	SMISC	2	15
Bending Moment *Mz*	SMISC	3	16
Torque *Mx*	SMISC	4	17
Shear Force in *XZ* Plane	SMISC	5	18
Shear Force in *XY* Plane	SMISC	6	19
Axial Strain	SMISC	7	20
Curvature *Kyy*	SMISC	8	21
Curvature *Kzz*	SMISC	9	22
Torsion curvature *Kxx*	SMISC	10	23
Transverse Shear Strain (*XZ*)	SMISC	11	24
Transverse Shear Strain (*XY*)	SMISC	12	25
Area of Cross Section	SMISC	13	26
Bimoment	SMISC	27	29
Bicurvature	SMISC	28	30

BEAM189 Example

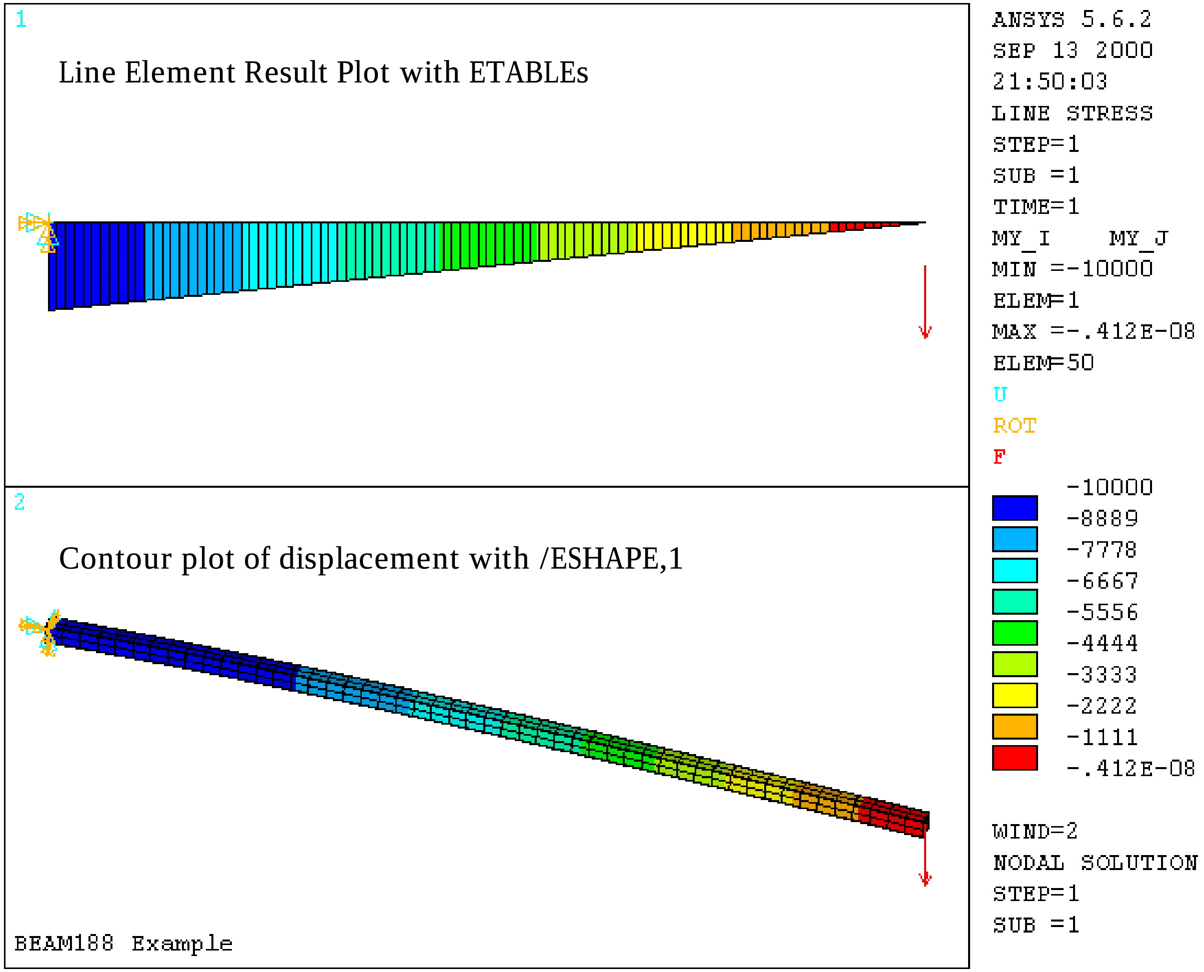
- After defining element tables for MY for node I and J, these are plotted as “Line Element Results” as noted below:



BEAM189 Example

Resulting line plot is shown at top-right. Allows for visualizing moment variation from nodes I to J for each element.

Force is 100, length is 100, so we expect max moment of 10000, as noted on right.



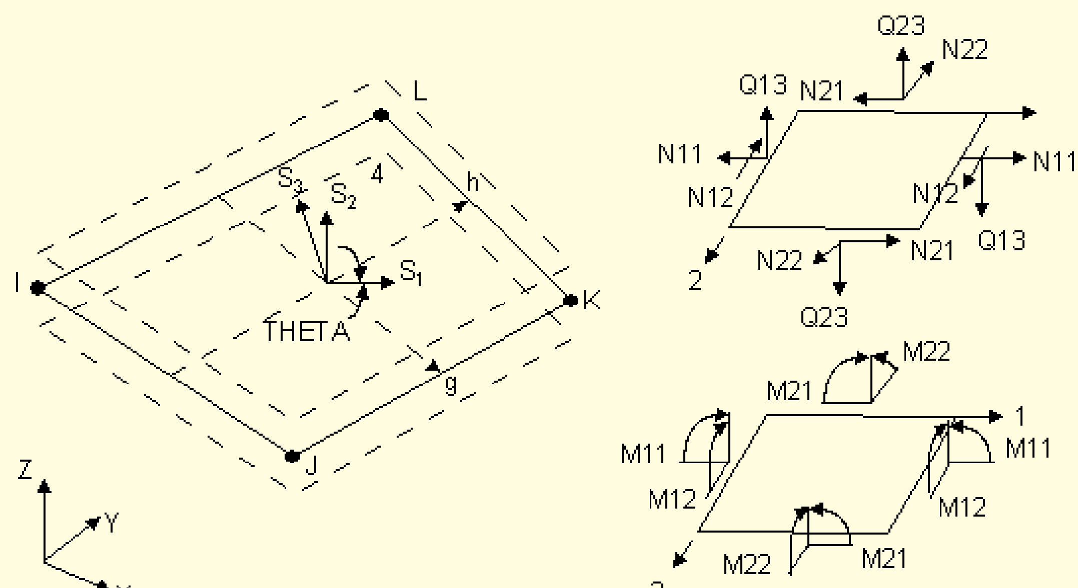
SHELL181 Example

- Stress resultants (force/length) can be obtained for SHELL181. Nij, Mij, and Qij are available as noted in the online help

The element stress resultants (N11, M11, Q13, etc.) are parallel to the element coordinate system, as are the membrane strains and curvatures of the element. Such generalized strains are available through the SMISC option at the element centroid only. The transverse shear forces Q13, Q23 are available only in resultant form: that is, use SMISC,7 (or 8). The element stress output will not therefore include the shear stress caused by transverse shear forces.

SHELL181 does not support basic element printout. POST1 provides more comprehensive output processing tools; therefore, we suggest using **OUTRES** to ensure that the required results are stored in the database.

Figure 3. SHELL181 Stress Output



SHELL181 Example

- In the online help, the “Sequence Numbers” for Nij, Mij, and Qij are listed for ETABLE definition purposes.

Table 3. SHELL181 Item and Sequence Numbers for the ETABLE and ESOL Commands

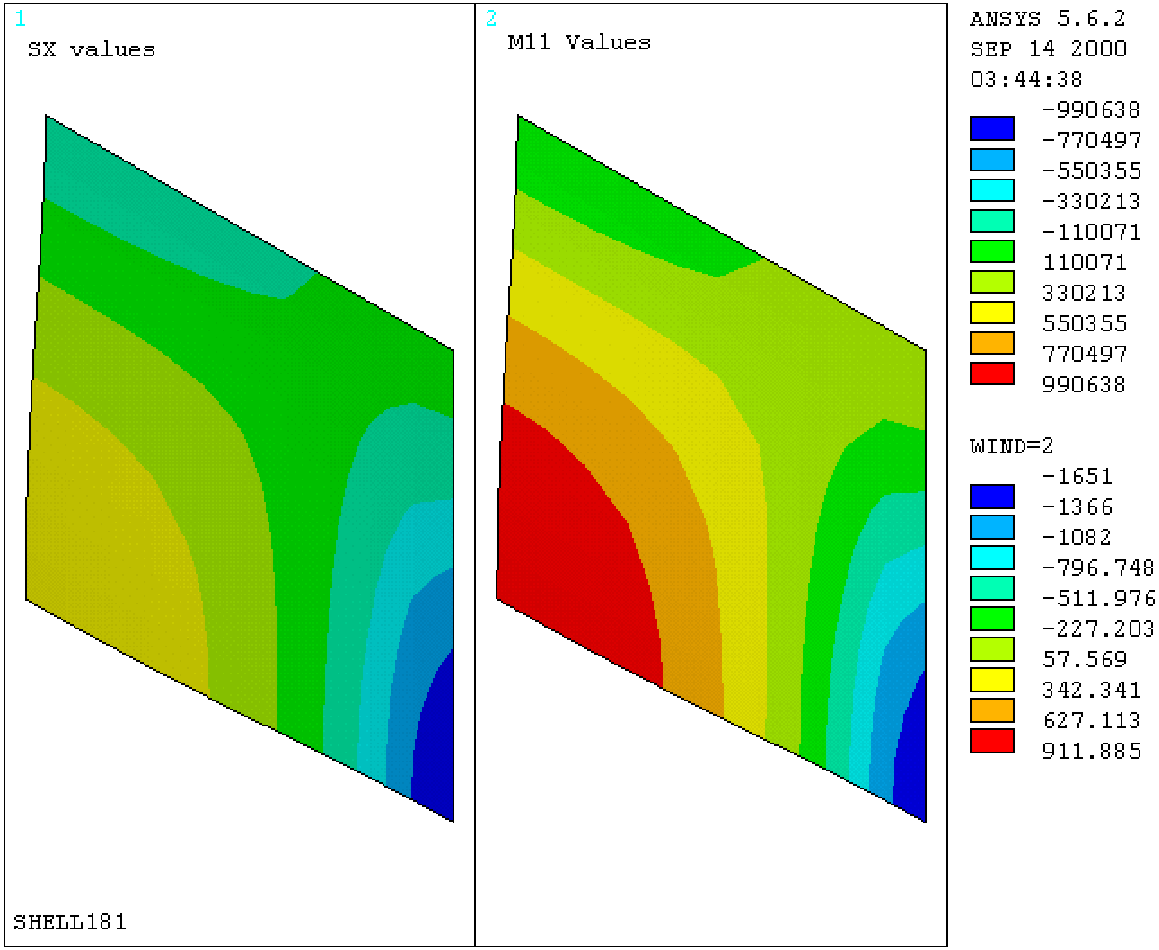
Name	Item	E	I	J	K	L
N11	SMISC	1	--	--	--	--
N22	SMISC	2	--	--	--	--
N12	SMISC	3	--	--	--	--
M11	SMISC	4	--	--	--	--
M22	SMISC	5	--	--	--	--
M12	SMISC	6	--	--	--	--
Q13[1]	SMISC	7	--	--	--	--
Q23[1]	SMISC	8	--	--	--	--
ε_{11}	SMISC	9	--	--	--	--
ε_{22}	SMISC	10	--	--	--	--
ε_{12}	SMISC	11	--	--	--	--
k_{11}	SMISC	12	--	--	--	--
k_{22}	SMISC	13	--	--	--	--
k_{12}	SMISC	14	--	--	--	--
γ_{13}	SMISC	15	--	--	--	--
γ_{23}	SMISC	16	--	--	--	--
THICK	SMISC	17	--	--	--	--
P1	SMISC	--	18	19	20	21

SHELL181 Example

Resulting plot of SX and M11 for a pressure-loaded plate is shown on right.

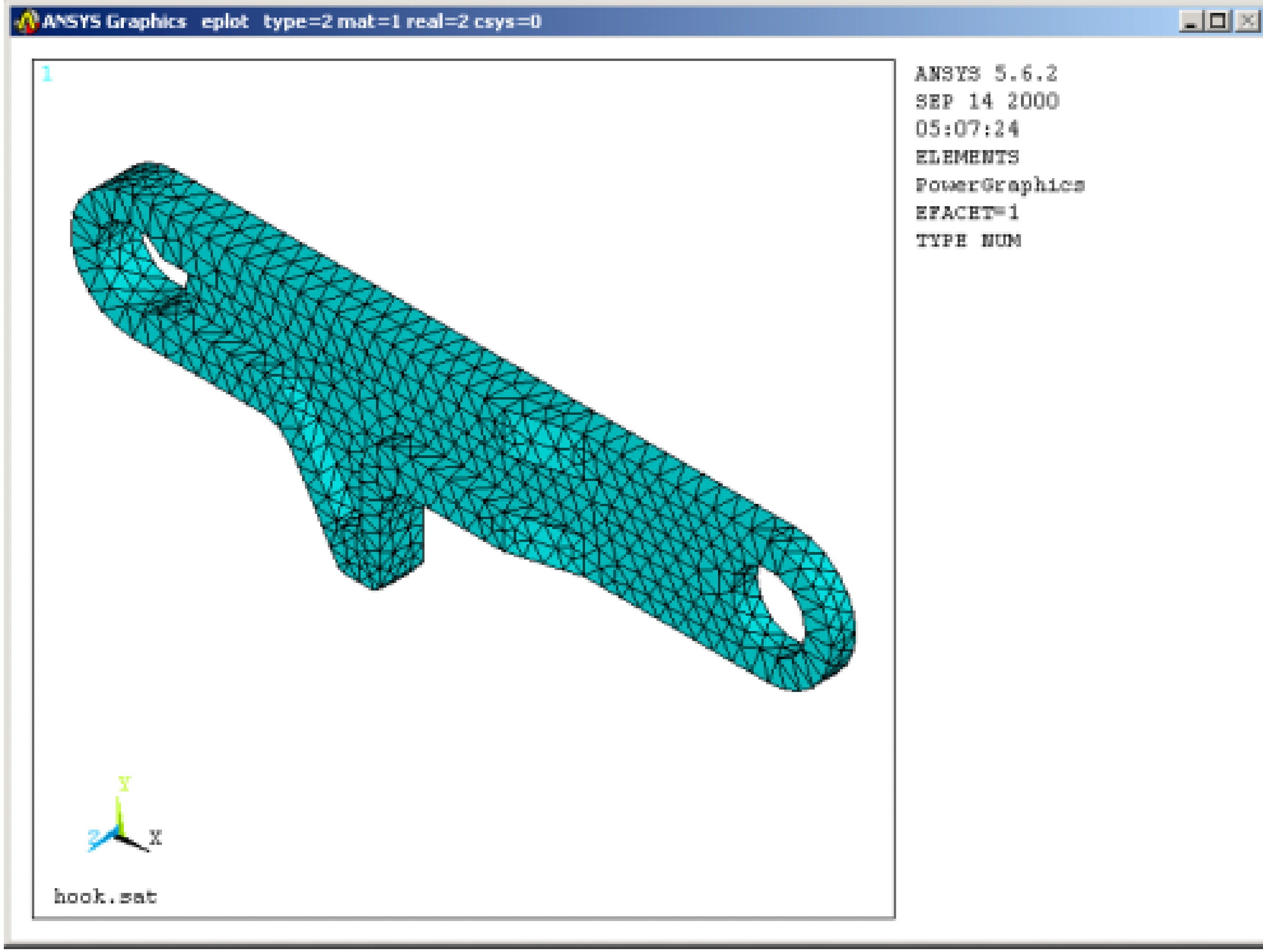
$SX=6*M11/t^2$

For this case, max SX is 990638. Thickness is 0.1, so we expect M11 to be 1651.06, which the obtained result.



SOLID92 Example

For a given mesh, we may want to calculate volumetric properties. This may also include 2D planar or axisymmetric elements. Element tables allow us to obtain volume of each element which we can then sum (SSUM command)



SOLID92 Example

- Compare solid model volume (VSUM) with volume obtained by summing each element's volume (SSUM)

VSUM Command

File

```
PRINT GEOMETRY ITEMS ASSOCIATED WITH THE CURRENTLY SELECTED VOLUMES

*** NOTE ***                            CP=      11.336    TIME= 05:06:11
Density not associated with all selected volumes.  Geometry items are
 based on a unit density.

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

  TOTAL VOLUME OF ALL SELECTED VOLUMES =   11.856

  CENTROID:  XC=-0.58395       YC=-0.14261       ZC= 0.28078

                    *** MOMENTS OF INERTIA ***
                     (BASED ON A UNIT DENSITY)

          ABOUT ORIGIN      ABOUT CENTROID      PRINCIPAL
    IXX =    8.9017            7.7259
    IYY =    108.35            103.37
    IZZ =    114.59            110.30
    IXY =   -2.5630           -1.5756
    IYZ =   0.49803           0.23270E-01
    IZX =    3.4105            1.4665

    PRINCIPAL ORIENTATION VECTORS (X,Y,Z):
      1.000  0.016 -0.014   -0.016  1.000  0.000
      (THXY=    0.943  THYZ=     0.007  THZX=     0
```

SSUM Command

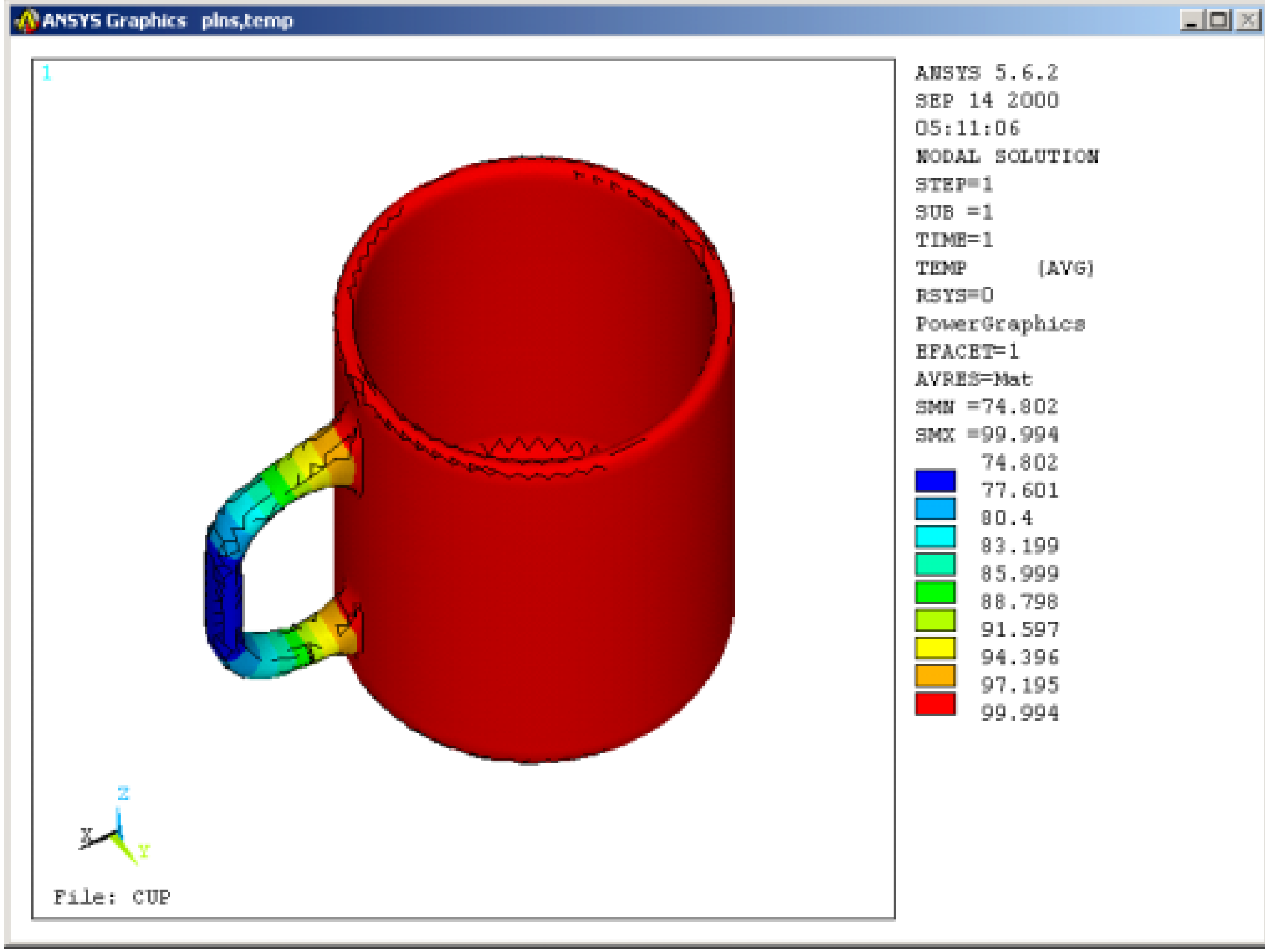
File

```
SUM ALL THE ACTIVE ENTRIES IN THE ELEMENT TABLE

TABLE LABEL     TOTAL
VOLUME      11.8567
```

SURF152 Example

Element tables of thermal solids or thermal surface effect elements can be used to obtain heat losses due to convection (and radiation). In this example, we will look at losses due to both modes of heat transfer for SURF154



SURF152 Example

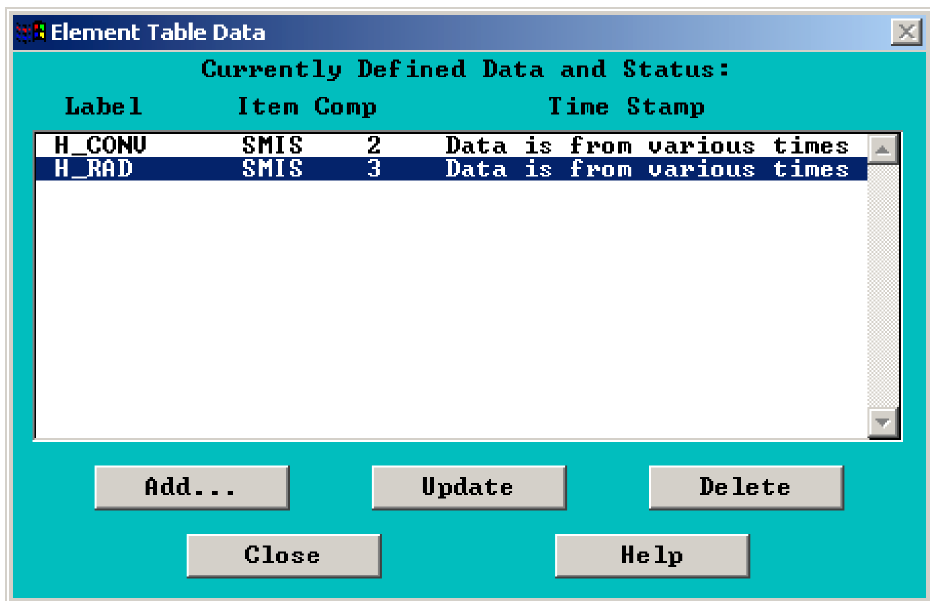
HGEN	HG(N), HG(O), HG(P)	2	--
HEAT GEN. RATE	Heat generation rate over entire element (HGTOT)	2	2
HFLUX	Heat flux at nodes I, J, K, L	3	
HEAT FLOW RATE	Heat flow rate over element surface area (HFCTOT)	3	
HFILM	Film coefficient at each face node	4	
TBULK	Bulk temperature at each face node or temperature of extra node	4	
TAVG	Average surface temperature	4	
TAW	Adiabatic wall temperature	5	
RELVEL	Relative velocity	5	
SPHTFL	Specific heat of the fluid	5	
RECFAC	Recovery factor	5	
CONV. HEAT RATE	Convection heat flow rate over element surface area (HFCTOT)	4	
CONV. HEAT RATE/AREA	Average convection heat flow rate per unit area	4	
EMISSUR	Average emissivity of surface (for element material number)	6	
EMISEXT	Emissivity of extra node	6	
TEMPSUR	Average temperature of surface	6	
TEMPEXT	Temperature of extra node	6	
FORM FACTOR	Average form factor of element	6	
RAD. HEAT RATE	Radiation heat flow rate over entire element (HRTOT)	6	
RAD. HEAT RATE/AREA	Average radiation heat flow rate per unit area	6	--

Table 2. SURF152 Item and Sequence Numbers for th

Name	Item	E
HGTOT	SMISC	1
HFCTOT	SMISC	2
HRTOT	SMISC	3
AREA	NMISC	1
VNX	NMISC	2
VNY	NMISC	3
VNZ	NMISC	4
HFILM	NMISC	5
TAVG	NMISC	6
TBULK	NMISC	7
TAW	NMISC	8
RELVEL	NMISC	9
SPHTFL	NMISC	10
RECFAC	NMISC	11
EMISSUR	NMISC	12

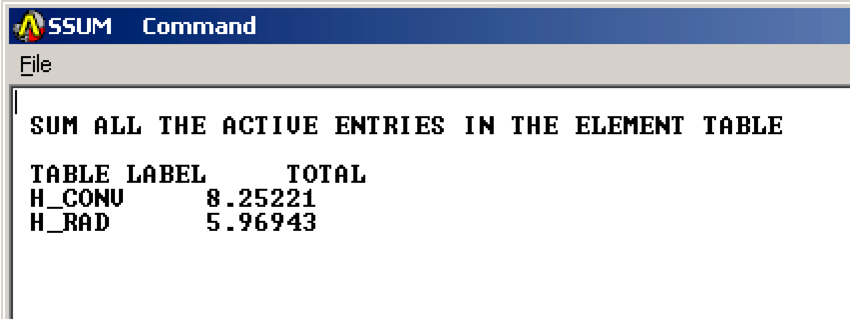
SURF152 Example

- Total heat flow rate is 14.2216, which can be confirmed with reaction force (heat flow).
- Define element tables for heat flow rate due to convection and radiation.
- Sum the values to obtain totals.



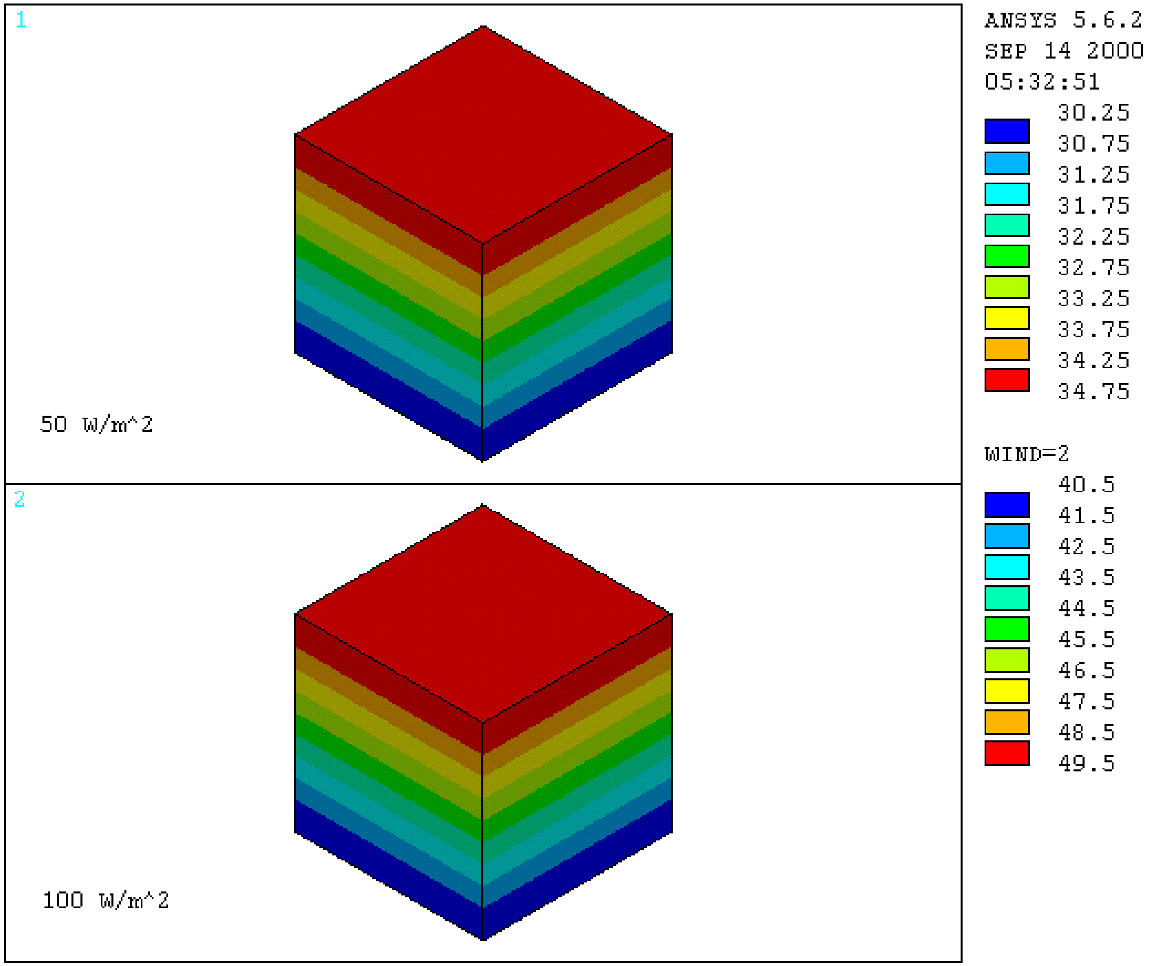
SURF152 Example

- Results noted below. Heat loss due to convection is 8.25, due to radiation is 5.97. This adds up to 14.22, as expected for heat balance.
- From the results, one can conclude that radiation is a significant heat path, as posed by the problem description. (One would have done thermal resistance network to confirm this beforehand)



Thermal Example

Element tables are static, so one can get element tables for different load cases (as shown on right) and compare results. With flux doubled, we expect ΔT doubled as well, which is what we get.



Conclusion

Additional Considerations

- Element tables are generally centroid values of elements. Some element tables can be a value at a given node, but only one value will be stored per element per ETABLE
  - For ETABLE results of DOF & other nodal quantities, these will be average of nodes of element, so min/max values will be reported lower than nodal values
- Parameter arrays can be inserted into ETABLE via *VPUT command
  - "Utility Menu > Parameters > Array Operations > Put Array Data…"
- Be sure to select the appropriate elements PRIOR to defining element tables, such that any calculations include only those pertinent element types
  - Some sequence items such as SMISC,2 or NMISC,5 mean different things for different elements. Including all elements may invalidate any further calculations.

Further References

- Ch. 5.2.3 “Creating an Element Table”, ANSYS 5.6 Basic Analysis Procedures Guide
- Ch. 2.2.2.2 “The ‘Item and Sequence Number’ Table”, ANSYS 5.6 Elements Reference
- “ETABLE”, ANSYS 5.6 Commands Reference