

Documentation for USER101: a 2D User-Defined Thermal Element in ANSYS

Lucas Brouwer

ATAP Division, Lawrence Berkeley National Laboratory

Documentation is given for a user defined 2D thermal element in ANSYS. This element is formulated similar to the standard distribution PLANE77, but is designed specifically for modeling conductor regions of superconducting magnets. Field, temperature, and critical current dependent material properties are implemented at the point of element matrix generation. This element can be used standalone for quench propagation studies, or in parallel with USER102 for coupled electromagnetic, circuit, and thermal simulation.

Contents

1	Element Form and DOF	3
2	Keyopts and Real Constants	4
3	Homoginization of Material Properties	7
4	Use with the Multi-field Solver	8
5	Element Output	10
6	Derivation of the Finite Element Matrices	11
7	Benchmarking and Verification	12
A	Known Limitations and Issues	13
B	Example: Element Mapping for MFS	14

1 Element Form and DOF

Similar to PLANE77, the higher order element has four midside and four corner nodes which are labeled as seen in Figure 1. Each node carries a single degree of freedom (temperature), which is interpolated throughout the element using the shape functions given in Equation 1. Integration of functions over the element, as needed for the generation of element matrices, is performed numerically using Gaussian quadrature with nine integration points. The location of the nine sampling points and their corresponding weights can be found in the ANSYS theory manual or introductory finite element textbooks [1, 2].

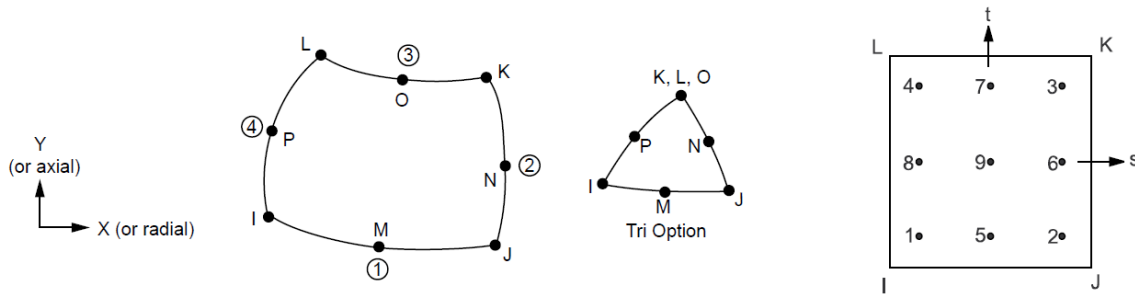


Figure 1: The ordering of the 8-node element geometry is shown on the left (Fig from [3], Pg. 283), and again for the nine Gaussian quadrature points on the right (Fig from [1], Pg. 358).

$$\begin{aligned}
 u = & + \frac{1}{4}u_I(1-s)(1-t)(-s-t-1) \\
 & + \frac{1}{4}u_J(1+s)(1-t)(+s-t-1) \\
 & + \frac{1}{4}u_K(1+s)(1+t)(+s+t-1) \\
 & + \frac{1}{4}u_L(1-s)(1+t)(-s+t-1) \\
 & + \frac{1}{2}u_M(1-s^2)(1-t) \\
 & + \frac{1}{2}u_N(1+s)(1-t^2) \\
 & + \frac{1}{2}u_O(1-s^2)(1+t) \\
 & + \frac{1}{2}u_P(1-s)(1-t^2)
 \end{aligned} \tag{1}$$

2 Keyopts and Real Constants

Similar to standard distribution elements, USER101 has several element key options used to define element properties. These are outlined in Table 1. Key option 1 specifies whether internal fits will be used for material properties (allowing for variation with temperature, magnetic field, and quench state), or whether the user will input properties in ANSYS format similar to PLANE77 (temperature variation only). If internal fits are chosen, key options 3-13 are used to select the materials present and desired fits. Key option 2 specifies whether the element will be coupled to the electromagnetic user element USER102 using the Multi-field Solver. This requires mapping of the element numbers between electromagnetic and thermal domains and also enables binary file writing for passing of non-standard Multi-field Solver variables. More details and additional set up steps for coupling between the user elements are outlined in Section 4.

Table 1: USER101 Key Options

Selection	Keyopt	Value	
Material Models ¹	(1)	0 1	use internal fits and homogenization (Sec. 3) ANSYS input for element cp, kxx, dens
Transfer with EMAG	(2)	0 1	no yes
Superconductor	(3)	0 1 2	NbTi (hold for future development) Nb ₃ Sn Bi2212 (hold for future development)
Stabilizer	(4)	0 1	Cu Ag (hold for future development)
Non-Cond. Material	(5)	0 1	G10 hold for future development
Cu cv fit	(6)	0 1 2 3	NIST CUDI Matpro userCucv.f (hold for future development)
Cu kxx fit	(7)	0 1 2 3	NIST CUDI Matpro userCukxx.f (hold for future development)
NbTi cv fit	(8)	0 1	hold for future development userNbTicv.f (hold for future development)
Nb ₃ Sn cv fit	(9)	0 1 2	NIST CUDI userNb3Sncv.f (hold for future development)
Bi2212 cv fit	(10)	0 1	hold for future development userBicv.f (hold for future development)
G10 cv fit	(11)	0 1	NIST userG10cv.f (hold for future development)
Ag cv fit	(12)	0 1	hold for future development userAgcv.f (hold for future development)
Ag kxx fit	(13)	0 1	hold for future development userAgkxx.f (hold for future development)

¹ for keyopt(1)=0, select materials and internal fits [4] using key options 3-13

The real constants in Table 2 are used to specify the geometry and properties of the modeled coil regions. In the case of coupled simulation with USER102 (keyopt(2)=1), the only real constant required is the element mapping number for the Multi-field Solver as explained in Section 4. In the case of stand alone use, other real constants are needed for material property evaluation and homogenization.

Table 2: USER101 Real Constants

Real Const.	Variable		Req. Keyopt
(1)	Emap	element number of mapped element in EMAG	(2)=1
(2)	Bev*	field for mat. prop.	(2)=0
(3)	qflag*	quench state for mat. prop.	(2)=0
(4)	RRR*	Stabilizer RRR for mat. prop. (Cu,Ag)	(2)=0
(5)	f_{cond}^*	fraction of conductor in coil region	(2)=0
(6)	f_{sc}^*	fraction of superconductor in conductor region	(2)=0

* These variables are mapped from EMAG with a binary file if keyopt(2)=1

3 Homoginization of Material Properties

The user element region is assumed to contain epoxy, copper stabilizer, and superconductor with the fraction of each given by the real constants f_{cond} and f_{sc} in Table 2. The material properties are blended using area fractions for thermal conductivity (kxx) and volumetric heat capacity (cv). Given the thermal conductivity of copper is orders of magnitude greater than the other components, a blend depending only on copper is used. The epoxy regions are considered to be glass fiber reinforced, and so the properties of G10 are assumed. In all cases the material properties are assumed to be isotropic. With these assumptions the blended properties used are given by

$$kxx = f_3(kxx_{Cu}) \quad (2)$$

and

$$cv = f_1(cv_{G10}) + f_2(cv_{Nb3Sn}) + f_2(cv_{Cu}) \quad (3)$$

where

$$\begin{aligned} f_1 &= 1 - f_{cond} \\ f_2 &= f_{cond}f_{sc} \\ f_3 &= f_{cond}(1 - f_{sc}). \end{aligned} \quad (4)$$

4 Use with the Multi-field Solver

The Multi-field Solver is a documented feature of ANSYS which allows for solving of sequentially coupled problems with independent meshes [5]. A unique, meshed region is generated for each physics field and load coupling interfaces for which loads will be passed between them are specified. Each region is solved independently with its own time stepping and solution options. The solver transfers the loads across the defined interfaces (even with dissimilar meshes), and iterates between each physics field in sequence until the transfer of loads converges for a user defined “stagger” time step as shown in figure 2.

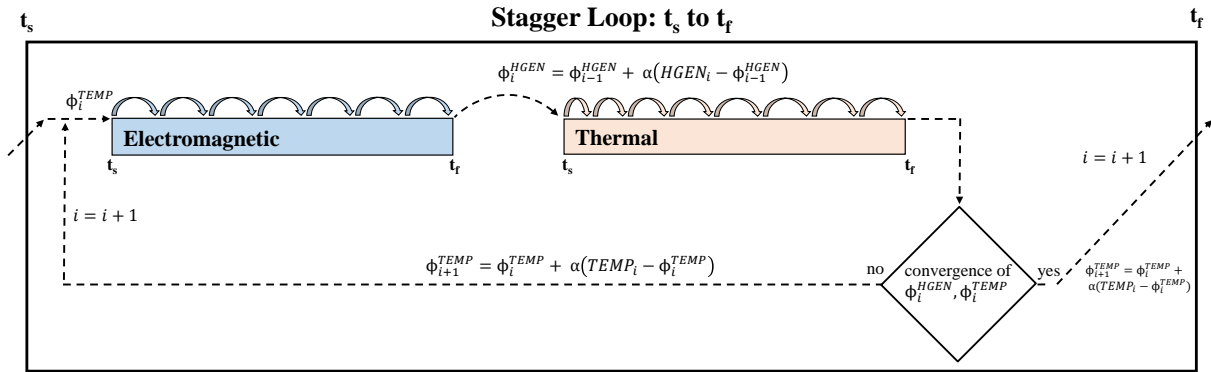


Figure 2: A stagger loop within the Multi-field Solver is shown for coupled electromagnetic and thermal fields (see figure 3 for an example of how such a simulation is set up with the user elements). In this example, the loads transferred between fields are heat generation ϕ^{HGEN} and temperature ϕ^{TEMP} . This approach loops over the stagger time step (from t_s to t_f) with a relaxation factor α applied to the load transfer until convergence of the loads is achieved. Separation of the problem into sequentially defined stagger steps is used to simulate over the entire time domain.

This solver has been successfully used for fully coupled simulations including the user elements. To do this, two physics fields are created which are shown labeled as “electromagnetic” and “thermal” in Figure 3. A load transfer interface is specified between meshed coil regions and any structural regions with eddy currents. This allows for passing Joule heat loads from the electromagnetic region to the thermal region, and passing temperature back. Both temperature and Joule heating are standard loads which may be transferred with the Multi-field Solver. To allow for thermal material properties to also vary with magnetic field and quench state, these variables are shared at every FEM iteration.

The method used for passing non-standard variables requires mesh element mapping be manually defined for user element regions. This is accomplished by using a similar mesh for matched regions between thermal and electromagnetic domains (AGEN is a useful command for this). Once a similar mesh has been created, the mapping is defined by setting real constant 1 of each thermal element as the matched element’s number in the electromagnetic region. An example of a simple APDL script performing this mesh generation and matching is given in appendix B. With the element numbers manually mapped, the transfer of non-

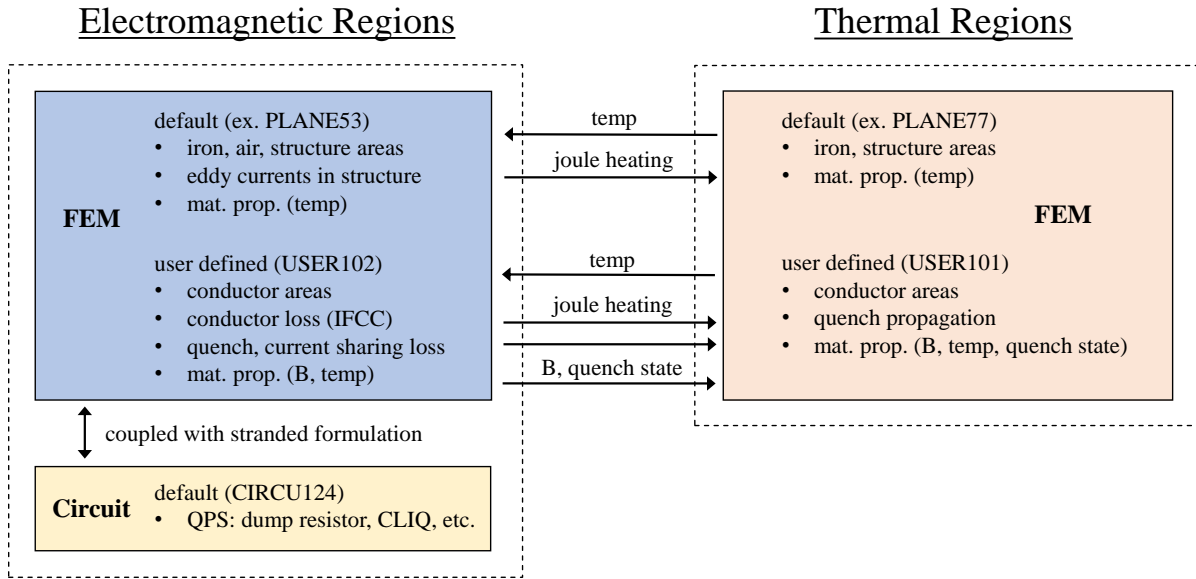


Figure 3: An overview of coupled thermal, electromagnetic, and circuit simulation in ANSYS with user defined elements is shown. Such an approach allows for simulating the impact of interfilament coupling currents, quench, and structural eddy currents on magnet behavior while including temperature and field dependent material properties. The independently meshed electromagnetic and thermal domains are solved concurrently using the Multi-field Solver.

standard MFS variables is performed during each element matrix generation.

5 Element Output

Table 3 lists the nodal and element results output to the ANSYS .db file. In addition to the standard results, a number of parameters are output as non-summable misc element results as described in Table 4. These can be plotted using “PLESOL,NMISC,#”, where “#” is the corresponding number in the table.

Table 3: USER101 Nodal and Element Output

Variable	Comp.	NSOL	ESOL	Description
temp		yes	no	temperature DOF
VOLU		no	yes	element area

Table 4: USER101 NMISC Output

NMISC	Variable	Description
(1)	T	elem temp used for mat. prop. etc.
(2)	Emap	element number of mapped element in EMAG
(3)	B	elem field used for mat. prop. etc.
(4)	qflag	(-1) = SC, (0) = current sharing, (1) = quenched
(5)	RRR	copper RRR
(6)	fcond	conductor fraction of modeled coil area
(7)	fsc	SC fraction of conductor
(8)	HGEN	elem heat generation
(9)	cvCu	copper heat capacity
(10)	kxxCu	copper thermal conductivity
(11)	cvNb3Sn	Nb ₃ Sn heat capacity
(12)	kxxNb3Sn	hold for future development
(13)	cvG10	G10 heat capacity
(14)	kxxG10	hold for future development
(15)	cv	homogenized heat capacity
(16)	kxx	homogenized thermal conductivity
(17)	dens	density if input using keyopt(1)=1

6 Derivation of the Finite Element Matrices

The formulation for the USER101 thermal element is identical to standard distribution PLANE77 element (new aspects are internal material property fits and coupling to USER102). The element matrices then match those given for PLANE77 in [\[1\]](#).

7 Benchmarking and Verification

Two verification and benchmarking studies have been performed. The first compares results for a single strand in a uniformly changing background field to analytic expectations. This study can be found in [6]. The second effort compares results from ANSYS to a similar implementation in COMSOL [7, 8] with the help of CERN. This extensive comparison can be found documented in [9].

A Known Limitations and Issues

Many of these are somewhat “easy” to implement or fix, but have been skipped over to get something up and running. There are also very few warnings set up for ill-defined input (mismatching keyopts, real constants, etc.) and unfortunately most problems will lead to a crash of ANSYS with no further output for debugging.

- General
 - the axisymmetric option is not yet included (contact lnbrouwer@lbl.gov if desired)
 - the Bi-2212 option is not yet included (contact lnbrouwer@lbl.gov if desired)
 - degenerate elements have issues (force quad mesh for user elem regions)
 - only compiled for single processor (run using SMP with 1 core)
 - no effort put into programming optimization for run time
 - no warnings if material property functions called beyond defined temperature range
- For Multi-field Solver (see Section 4 for more details)
 - marked load transfer regions must have similar mesh
 - thermal elements must be flagged for load transfer using real const
- USER101
 - recommended use is for conductor regions only (use PLANE77 elsewhere)
 - loading outside of HGEN (on lines etc.) may have issues
- USER102
 - recommended use is for conductor regions only (use PLANE53 elsewhere)
 - Lorentz forces are not yet included in the output

B Example: Element Mapping for MFS

This APDL script demonstrates how to generate and manually map a thermal domain from a previously created electromagnetic domain for the Multi-field Solver (see Section 4). Prior to this script, a group of several areas named “cond” has been created and meshed with USER102 (electromagnetic) elements. This script generates a new meshed area for each area in “cond”, and sets real constant 1 of the new mesh elements to match the corresponding element number in the original electromagnetic region. The new regions are then grouped under the name “tcond” and the new mesh is modified to the thermal user element USER101.

```

/***** Strand Element Mapping from Magnetic to Thermal *****/
/*****

!match elements with a constant shift instead (no loop - much faster)
alls
numcmp,elem

cmsel,s,cond
alls,below,area
*get,nna,area,,count
*do,j,1,nna
  *get,na%%j,area,0,nxth
  asel,u,area,,na%%j
*enddo

! starting number for the real constants -> make sure no overlap with previously defined real constants
cnt = 500 !this needs to be greater than all other reals
*do,j,1,nna
  asel,s,area,,na%%j
  alls,below,area
  *get,enum,elem,,count
  *get,eMi,elem,,num,min

  agen,2,all,,,,,0
  asel,u,area,,na%%j
  cm,tcond%%j,area

  cmsel,s,tcond%%j
  alls,below,area
  *get,eTi,elem,,num,min

  *do,i,1,enum
    r,cnt,eMi+(i-1)
    ettt = eTi+(i-1)
    esel,s,elem,,ettt
    emodif,all,real,cnt
    cnt = cnt + 1
  *enddo
*enddo

cmsel,s,tcond1
*do,j,2,nna
  cmsel,a,tcond%%j
*enddo
alls,below,area
cm,tcond,area

/***** Define EMAG/THERMAL Interfaces *****/
/*****

cmsel,s,cond
cmsel,a,tcond
alls,below,area
bfe,all,fvin,,1 !define interfaces on both elements

/***** Change Thermal Region Mesh to USER101 *****/
/*****

! thermal for conductor region only
et,12,user101
keyopt,12,1,0 ! 0=internal fits, 1=ANSYS table
keyopt,12,2,1 ! 0=no transfer to mag, 1=transfer to mag

cmsel,s,tcond
alls,below,area
emodif,all,mat,11
emodif,all,type,12

```

References

- [1] “ANSYS Mechanical APDL Theory Reference, Release 17.1,” 2016.
- [2] K. J. Bathe, *Finite Element Procedures in Engineering Analysis*. Prentice-Hall, 1982.
- [3] “ANSYS Mechanical APDL Element Reference, Release 17.1,” 2016.
- [4] G. Manfreda, “Review of ROXIE’s Material Properties Database for Quench Simulation,” *CERN Internal Note: EDMS NR*, vol. 1178007, 2018.
- [5] “ANSYS Mechanical APDL Coupled-Field Analysis Guide, Release 17.1,” 2016.
- [6] L. Brouwer, “Check of IFCC Equivalent Magnetization Implementation in ANSYS User Elements using a Single Strand Model,” *LBNL Eng. Note: SU-1010-4842, R1.0*, 2019. [Online]. Available: <https://usmdp.lbl.gov/scpack-code/>
- [7] L. Bortot, B. Auchmann, I. Cortez-Garcia, A. M. Fernandez-Navarro, M. Maciejewski, M. Mentink, M. Prioli, E. Ravaioli, S. Schops, and A. Verweij, “Steam: A hierarchical co-simulation framework for superconducting accelerator magnet circuits,” *IEEE Trans. Appl. Supercond.*, vol. 28, no. 3, p. 4900706, 2018.
- [8] L. Bortot, B. Auchmann, I. Cortez-Garcia, A. M. Fernandez-Navarro, M. Maciejewski, M. Prioli, S. Schops, and A. Verweij, “A 2-d finite-element model for electro-thermal transients in accelerator magnets,” *IEEE Trans. Appl. Supercond.*, vol. 54, no. 3, p. 7000404, 2018.
- [9] L. Brouwer, B. Auchmann, L. Bortot, and E. Stubberud, “Crosscheck of the ANSYS-COMSOL 2D FEM Implementations for Superconducting Accelerator Magnets,” *LBNL Eng. Note: SU-1010-4841, R1.0*, 2019. [Online]. Available: <https://usmdp.lbl.gov/scpack-code/>