UCRL-ID--104090

DE91 000073

REMAP: a computer code that transfers node information between dissimilar grids

Arthur B. Shapiro

Lawrence Livermore National Laboratory

April 1990

Lawrence Livermore National Laboratory
Mail Stop L-194
7000 East Ave.
Livermore, CA 94550
phone: 415-422-5066
fax: 415-422-1370

DISTRIBUTION OF THIS DOCUMENT IS UNLIMITED
MASTER *eb*

Table of Contents

1	Introduction	1
2	Execution and Input	3
2.1	Terminal Execution Line	3
2.2	Input File	4
2.2.1	Control Card	4
2.2.2	Node Point Data	5
2.2.3	Element Data	7
3	Example	9
3.1	REMAP Example	9
3.2	WARNING - User Beware	15
4	Theory	17
4.1	Coarse Filter	17
4.2	Fine Filter	18
4.3	Shape Functions	20
4.4	Calculating the Natural Coordinates	22
4.5	Temperature Interpolation	23
	References	25

1 *Introduction*

REMAP is a computer code that transfers the axisymmetric, two dimensional planar, or three dimensional temperature field from one finite element mesh to another. The meshes may be arbitrary as far as the number of elements and their geometry. REMAP interpolates or extrapolates the node temperatures from the old mesh to the new mesh using linear, bilinear, or trilinear isoparametric finite element shape functions. REMAP is used to transfer the temperature field from a thermal analysis mesh to a more finely discretized structural analysis mesh when performing a thermal stress analysis. REMAP was designed to be used with the finite element heat transfer codes TOPAZ2D [1] and TOPAZ3D [2], and the solid mechanics codes NIKE2D [3] and NIKE3D [4]. The I/O formats in REMAP can be easily modified to accept input from other codes (e.g., finite difference) and generate output files for other structural codes. REMAP can be used to transfer any scalar field variable between dissimilar finite element meshes.

Three papers by Thomson from NASA Lewis and Maffeo from GE on the TRANCITS code [5-7] were found useful in developing REMAP. Their idea of a coarse filter followed by a fine filter to determine which element from the old mesh contains a node point from the new mesh was used. The coarse filter determines a subset of elements from the old mesh that may contain the new node point. The fine filter determines the element that contains the new node point. REMAP uses the ray-surface intersection algorithm developed for the FACET [8] code for the fine filter. This algorithm has the added capability to determine which element the node is closest to if the node point lies outside the perimeter (or surface) of the old mesh. Once an element from the old mesh has been identified as containing or closest to the new node point, the natural coordinates for the node point are calculated. The isoparametric finite element shape functions are calculated next. These shape functions are then used to interpolate or extrapolate the temperatures from the nodes comprising the old element to the new node point.

Chapter 2 contains information on preparing an input file and running REMAP. Example problems are presented in Chapter 3. The mathematical theory and numerical implementation are presented in Chapter 4.

2 *Execution and Input*

2.1 Terminal Execution Line

REMAP is kept in the public library MDGLIB on all Cray machines. To obtain a copy of REMAP in your private disk space, type

exe mdglib remap x.

The terminal execution line for REMAP is

remap i=inf g=tpf o=nkf p=pfl

where

- inf=text input file containing control information, new node point coordinates, and new element connectivity.
- tpf=absolute input file containing old node point coordinates, temperatures, and old element connectivity. This is a TOPAZ plot file created from running TOPAZ for the thermal analysis. If you do not need to remap, then this file is the temperature input file to be used with NIKE for a thermal stress analysis.
- nkf=absolute output file containing the new node point coordinates, temperatures, and element connectivity. This is the temperature input file to be used with NIKE for a thermal stress analysis on the new mesh. (default - nkplot)
- pfl=text output file listing the input information and temperatures at the node points of the new mesh. (default=printfl)

File name dropouts are permitted, for example

remap i=inf g=tpf

The two output files, nkf and pfl, will be created with the default names nkplot and printfl, respectively.

2.2 Input File

NOTE: Values entered are for the new mesh

2.2.1 Control Card

<i>column</i>	<i>format</i>	<i>quantity</i>
1-5	I5	number of materials
6-10	I5	number of node points
11-15	I5	number of elements
16-20	I5	geometry type (1=axisymmetric, 2=2D planar, 3=3D)
21-25	I5	temperature flag (0=default; 1=use temperatures on the nodal cards as the initial condition)

2.2.2 Node Point Data

The node point data is formatted according to NIKE2D or NIKE3D. Supply one card for each node point with the following information:

NIKE2D - Axisymmetric and 2D-planar Mesh

<i>column</i>	<i>format</i>	<i>quantity</i>
1-5	I5	node point number, n^i
6-10	F5.0	displacement boundary condition code, (see NIKE2D users manual [3])
11-20	E10.0	r^i coordinate
21-30	E10.0	z^i coordinate
31-35	I5	node generation increment, k
36-45	E10.0	node point temperature - if not equal to zero, then this temperature will overwrite the initial temperature in the TOPAZ temperature file

Node point cards do not need to be in order. However, the highest node point number must terminate the node data. When node data is missing, node numbers are generated according to the sequence

$$n^i, n^i+k, n^i+2k, \dots, n^j$$

where n^i and n^j are the node numbers defined on two consecutive cards, and k is taken from the first card. Linear interpolation is used to obtain the coordinate of the generated nodes and the node temperatures. The boundary condition code of generated data is set to zero whenever the boundary condition code of n^i differs from that of n^j .

NIKE3D - 3 Dimensional Mesh

<i>column</i>	<i>format</i>	<i>quantity</i>
1-5	I5	node point number, n^i
6-10	I5	displacement boundary condition code (see NIKE3D users manual [4])
11-30	E20.0	x^i coordinate
31-50	E20.0	y^i coordinate
51-70	E20.0	z^i coordinate
71-75	I5	node generation increment, k
76-80	I5	rotational boundary condition code (see NIKE3D users manual [4])

2.2.3 Element Data

The element data is formatted according to NIKE2D or NIKE3D. Supply one card for each element with the following information:

NIKE2D - Axisymmetric and 2D-planar Mesh

<i>column</i>	<i>format</i>	<i>quantity</i>
1-5	I5	element number
6-10	I5	node n_1
11-15	I5	node n_2
16-20	I5	node n_3
21-25	I5	node n_4
26-30	I5	material number
31-35	I5	element node point generation increment, k
36-80		Data in these columns are not read by REMAP. However, this information may be required for the NIKE2D simulation.

Element cards are assumed to be in element number sequence. The highest element number must terminate the data. Omitted data is automatically generated with respect to the first card prior to the omitted data by $n^{i+1} = n^i + k$. The material property number and element node point generation increment are taken from the first card. The convention for numbering node points is shown below.

NIKE3D - 3 Dimensional Mesh using 4, 6, or 8 node solid elements

column	format	quantity
1-5	I5	element number
6-10	I5	material number
11-15	I5	element node point generation increment
16-20	I5	node n_1
21-25	I5	node n_2
26-30	I5	node n_3
31-35	I5	node n_4
36-40	I5	node n_5
41-45	I5	node n_6
46-50	I5	node n_7
51-55	I5	node n_8

The convention for numbering 4, 6, and 8 node solid elements are shown below.

3 Example

3.1 REMAP Example

The only intent of the following hypothetical problem is to demonstrate the procedure in using REMAP. There is no physical basis to the problem definition. The objective is to perform a thermal analysis on one mesh and then map the temperature field onto another mesh for the stress analysis. The TOPAZ2D input file for this problem is shown in Fig 1. The input file defines a 2-dimensional slab, 2 units long by 0.6 units thick, with temperature boundary conditions of 0F and 2F applied to the vertical end surfaces. The lateral surfaces are adiabatic. The material density, heat capacity, and thermal conductivity are all given a value of unity. The finite element mesh is shown in Fig 2 and isotherms of the calculated node point temperatures are shown in Fig 3. The post processing code ORION [10] was used to generate the mesh and isotherm plots using the output plot file, T2PLOT, created by TOPAZ2D.

```
1  example problem
2      1      6      2      2
3              4
4      0
5              0.      1.      1.
6      0
7  mat11      1      1      1.
8              1.      1.
9              1.
10             0.
11             0.
12             0.
13      1              0.      .2      0.      2
14      5              2.      .2      0.
15      2              0.      .8      0.      2
16      6              2.      .8      0.
17      1      1      3      4      2      1      1      2
18      1              1      1      0      0.
19      5              1      1      0      2.
```

FIGURE 1: TOPAZ2D input file defining the heat transfer problem.

Example

FIGURE 2: The thermal mesh consists of 2 elements and 6 node points.

FIGURE 3: Shown is the steady state temperature field. The isotherms are vertical because the heat flow is 1-dimensional from the right surface to the left surface. Also note that the temperature gradient varies linearly between the end surface boundary conditions of 0F and 2F.

Example

The REMAP input file, REMAPIN, defining the new mesh for the stress problem is shown in Fig 4. The execution line for REMAP using the files REMAPIN which defines the stress mesh, and T2PLOT which defines the thermal mesh is

remap i=remapin g=t2plot / t v

REMAP creates a file with the default name NKPLOT. This contains the new mesh data and remapped temperatures to be used as the temperature input file to NIKE2D. ORION can be used to display the new mesh (Fig 5) and isotherms of the remapped temperatures (Fig 6). I advise using ORION to plot isotherms on the new mesh and comparing them to isotherms on the old mesh to see if they make sense before performing the thermal stress simulation.

1	1	6	2	2	0		
2	1			0.		.0	2
3	5			2.		.4	
4	2			0.		1.	2
5	6			2.		.6	
6	1	1	3	4	2	1	
7	2	3	5	6	4	1	

FIGURE 4: The REMAP input file defining the stress mesh.

FIGURE 5: The new mesh to be used for the stress problem. The temperatures at node points 1 and 2 will require extrapolation from the thermal mesh (Fig 2), while temperatures at node points 3, 4, 5, and 6 will require interpolation.

Example

FIGURE 6: Shown is the steady state temperature field mapped onto the stress mesh using the temperature field data from the thermal mesh (Fig 3).

3.2 WARNING - User Beware

The mapping of the temperature field from the thermal mesh (Fig 2) to the stress mesh (Fig 5) is done by interpolation or extrapolation. The governing physics is ignored. Two dimensional heat transfer must be considered to accurately calculate the temperature field for the drastically different shape of the stress mesh. Figure 7 shows the isotherms on the stress mesh when 2-dimensional effects are considered. In comparing Fig 7 with Fig 6, note that the isotherms are curved and are shifted to the right. Although this is an extreme example, you should be aware of the physics of the problem that is being remapped.

FIGURE 7: Shown are the steady state isotherms on the stress mesh when 2-dimensional heat transfer is considered. Note the difference in shape and position of the isotherms between this figure and Fig 6.

4 *Theory*

REMAP transfers information (e.g, node point temperature) from an old mesh to a new mesh. To accomplish this, the element from the old mesh containing or closest to a node from the new mesh must be identified. This is achieved by an element search comprised of a coarse filter followed by a fine filter. The coarse filter uses a simple distance algorithm that identifies a subset of candidate elements. The fine filter determines the one element that contains or is closest to the new node point. Next, the finite element natural coordinates for the new node point are calculated with reference to the element, followed by the calculation of the isoparametric shape functions. Using these shape functions and the values of the field variable at the element's node points, the value at the new node point is calculated.

Following is a detailed description of the above procedure for two dimensional planar meshes. The two dimensional planar results are directly applicable to an axisymmetric mesh. Only the final equations are given for 3-dimensional meshes. For ease of presentation, the following terminology will be followed:

- **element** refers to the finite elements that comprise the old mesh
- **stress point** refers to the node points that comprise the new mesh

4.1 Coarse Filter

For every stress point (i.e., node point) in the new mesh, the coarse filter searches through all the elements in the old mesh to determine a subset of elements that may contain the stress points. A simple coordinate location check is performed. First, the extreme values of the node point coordinates comprising the element are determined. For a 4 node quadrilateral

$$\begin{aligned}x_{min} &= \min(x_1, x_2, x_3, x_4) \\x_{max} &= \max(x_1, x_2, x_3, x_4) \\y_{min} &= \min(y_1, y_2, y_3, y_4) \\y_{max} &= \max(y_1, y_2, y_3, y_4)\end{aligned}\tag{EQ 1}$$

Next the stress point coordinates (x_s, y_s) are compared to the extreme values of the node points. The stress point is contained by the element if

$$\begin{aligned} x_{min} &\leq x_s \leq x_{max} \\ y_{min} &\leq y_s \leq y_{max} \end{aligned} \quad (\text{EQ 2})$$

If the (EQ 2) relationships are satisfied, then the element is included in the subset of possible elements that contain the stress point. As shown in Fig 8, the (EQ 2) conditions are necessary but not sufficient to identify the one element that contains the stress point.

FIGURE 8: A stress point satisfying (EQ 2) but located outside of the element

The z-coordinate must be included in (EQ 1) and (EQ 2) for three dimensional meshes. Additionally, the coordinates of all the element nodes (i.e., 6 for a wedge, 8 for a hexahedral) must be included in determining the extreme values by (EQ 1).

4.2 Fine Filter

The fine filter searches through the subset of elements identified by the coarse filter to find the one element that contains the stress point. The symbols used in the following equations are defined in Fig 9.

$$\begin{aligned} V_s &= V_1 + \alpha_1 V_{12} + \alpha_2 V_{14} \\ V_s &= V_3 + \alpha_3 V_{32} + \alpha_4 V_{34} \end{aligned} \quad (\text{EQ 3})$$

FIGURE 9: This sketch illustrates the symbols used in (EQ 3)

Each of these vector equations can be expanded in terms of their components. This provides the following two sets of simultaneous equations to be solved for α_1 , α_2 , α_3 , and α_4 .

$$\begin{bmatrix} x_2 - x_1 & x_4 - x_1 \\ y_2 - y_1 & y_4 - y_1 \end{bmatrix} \begin{bmatrix} \alpha_1 \\ \alpha_2 \end{bmatrix} = \begin{bmatrix} x_s - x_1 \\ y_s - y_1 \end{bmatrix} \quad (\text{EQ 4})$$

$$\begin{bmatrix} x_2 - x_3 & x_4 - x_3 \\ y_2 - y_3 & y_4 - y_3 \end{bmatrix} \begin{bmatrix} \alpha_3 \\ \alpha_4 \end{bmatrix} = \begin{bmatrix} x_s - x_3 \\ y_s - y_3 \end{bmatrix}$$

If α_1 , α_2 , α_3 , and α_4 are greater than or equal to zero, then the stress point is within or on the surface of the element. A way of mentally visualizing this algorithm is that if the stress point is within both of the angles $\angle 412$ and $\angle 234$, then it is within the element. This algorithm is easily extended into three dimensions. The vector equations for an 8 node hexahedral element (Fig 10) are:

$$\begin{aligned} \mathbf{V}_s &= \mathbf{V}_1 + \alpha_1 \mathbf{V}_{12} + \alpha_2 \mathbf{V}_{14} + \alpha_3 \mathbf{V}_{15} \\ \mathbf{V}_s &= \mathbf{V}_3 + \alpha_4 \mathbf{V}_{37} + \alpha_5 \mathbf{V}_{34} + \alpha_6 \mathbf{V}_{32} \\ \mathbf{V}_s &= \mathbf{V}_6 + \alpha_7 \mathbf{V}_{62} + \alpha_8 \mathbf{V}_{65} + \alpha_9 \mathbf{V}_{67} \\ \mathbf{V}_s &= \mathbf{V}_8 + \alpha_{10} \mathbf{V}_{85} + \alpha_{11} \mathbf{V}_{84} + \alpha_{12} \mathbf{V}_{87} \end{aligned} \quad (\text{EQ 5})$$

Each of these equation can be expanded in terms of their components providing 12 simultaneous equations for the 12 unknown α 's. If all α 's are greater than or equal to zero, then the stress point is within or on the surface of the element. Similar algorithms for a 3 node triangle, a 4 node tetrahedral, and a 6 node wedge are included in REMAP.

If the fine filter does not find an element that contains the stress point, then the stress point lies outside the subset of elements. The closest element to the stress point is then determined. The sum of the distances between the stress point and the element nodes is calculated. The element for which this sum is a minimum is selected as the element closest to the stress point.

FIGURE 10: Node point numbering for a hexahedral element

4.3 Shape Functions

The domain of a straight edged quadrilateral element is defined by the locations of its four node points $\{(x_a, y_a), a=1, \dots, 4\}$. We seek a change of coordinates which maps the given quadrilateral into the biunit square, as depicted in Fig 11. The coordinates (ξ, η) are sometimes called the "natural coordinates". The coordinates (ξ, η) of a point in the biunit square are related to the coordinates (x, y) by mappings of the form (see Hughes [9])

$$\begin{aligned} x(\xi, \eta) &= \sum_{a=1}^4 N_a(\xi, \eta) x_a^e \\ y(\xi, \eta) &= \sum_{a=1}^4 N_a(\xi, \eta) y_a^e \end{aligned} \quad (\text{EQ 6})$$

The functions N_a are the finite element shape functions. For a bilinear quadrilateral they are

$$\begin{aligned} N_1 &= \frac{1}{4} (1 - \xi) (1 - \eta) & N_3 &= \frac{1}{4} (1 + \xi) (1 + \eta) \\ N_2 &= \frac{1}{4} (1 + \xi) (1 - \eta) & N_4 &= \frac{1}{4} (1 - \xi) (1 + \eta) \end{aligned} \quad (\text{EQ 7})$$

FIGURE 11: Mapping of a quadrilateral into a biunit square

The node numbering for a triangle, 4 node tetrahedral, and 6 node wedge are shown in Fig 12. The node numbering for an 8 node hexahedral has previously been shown in Fig 10. The shape functions for a linear triangle are

$$N_1 = \xi \quad N_2 = \eta \quad N_3 = 1 - \xi - \eta \quad (\text{EQ 8})$$

The shape functions for a linear 4 node tetrahedral are

$$\begin{aligned} N_1 &= \xi & N_3 &= \zeta \\ N_2 &= \eta & N_4 &= 1 - \xi - \eta - \zeta \end{aligned} \quad (\text{EQ 9})$$

The shape functions for a bilinear 6 node wedge are

$$\begin{aligned} N_1 &= \frac{1}{2} (1 - \zeta) \xi & N_4 &= \frac{1}{2} (1 + \zeta) \xi \\ N_2 &= \frac{1}{2} (1 - \zeta) \eta & N_5 &= \frac{1}{2} (1 + \zeta) \eta \\ N_3 &= \frac{1}{2} (1 - \zeta) (1 - \xi - \eta) & N_6 &= \frac{1}{2} (1 + \zeta) (1 - \xi - \eta) \end{aligned} \quad (\text{EQ 10})$$

The shape functions for a trilinear 8 node hexahedral are

$$\begin{aligned}
 N_1 &= \frac{1}{8} (1 - \xi) (1 - \eta) (1 - \zeta) & N_5 &= \frac{1}{8} (1 - \xi) (1 - \eta) (1 + \zeta) \\
 N_2 &= \frac{1}{8} (1 + \xi) (1 - \eta) (1 - \zeta) & N_6 &= \frac{1}{8} (1 + \xi) (1 - \eta) (1 + \zeta) \\
 N_3 &= \frac{1}{8} (1 + \xi) (1 + \eta) (1 - \zeta) & N_7 &= \frac{1}{8} (1 + \xi) (1 + \eta) (1 + \zeta) \\
 N_4 &= \frac{1}{8} (1 - \xi) (1 + \eta) (1 - \zeta) & N_8 &= \frac{1}{8} (1 - \xi) (1 + \eta) (1 + \zeta)
 \end{aligned}
 \tag{EQ 11}$$

FIGURE 12: Node point numbering for a triangle, tetrahedral, and wedge

4.4 Calculating the Natural Coordinates

For a quadrilateral, the natural coordinates corresponding to the stress point can be found by expanding (EQ 6) in terms of the shape functions (EQ 7), element node point coordinates, and stress point coordinates. This provides two nonlinear simultaneous equations that can be solved by Newton's method for ξ and η .

$$\begin{aligned}
 4x_s &= (x_1 + x_2 + x_3 + x_4) - (x_1 - x_2 - x_3 + x_4) \xi \\
 &\quad - (x_1 + x_2 - x_3 - x_4) \eta + (x_1 - x_2 + x_3 - x_4) \xi \eta \\
 4y_s &= (y_1 + y_2 + y_3 + y_4) - (y_1 - y_2 - y_3 + y_4) \xi \\
 &\quad - (y_1 + y_2 - y_3 - y_4) \eta + (y_1 - y_2 + y_3 - y_4) \xi \eta
 \end{aligned}
 \tag{EQ 12}$$

Note that (EQ 12) can be solved analytically. However, in the computer implementation of the exact solution a decision must be made as to which of the 4 roots should be kept. This decision is easy if the stress point is within the element. Then the roots with values in the range $[-1, 1]$ are retained. No clear decision can be made if the stress point is outside the element. Both the exact solution and Newton's method were coded. Newton's method was found to be more reliable in determining the roots of (EQ 12).

The procedure for the linear triangle, linear tetrahedral, 6 node wedge, and 8 node hexahedral are similar.

4.5 Temperature Interpolation

The shape functions chosen for the coordinate transformations (EQs 7-11) can be used to interpolate or extrapolate the element node point temperatures to the stress point (i.e., isoparametric concept) by

$$T_s = \sum_{a=1}^n N_a T_a^e \quad \text{where} \quad \begin{array}{ll} n = 3 & \text{triangle} \\ n = 4 & \text{quadrilateral} \\ n = 6 & \text{tetrahedral} \\ n = 8 & \text{hexahedral} \end{array} \quad (\text{EQ 13})$$

References

- 1 A. B. Shapiro, "TOPAZ2D - A Two Dimensional Finite Element Code for Heat Transfer Analysis, Electrostatics, and Magnetostatic Problems", University of California, Lawrence Livermore National Laboratory, Rept UCID-20824, 1986.
- 2 A. B. Shapiro, "TOPAZ3D - A Three Dimensional Finite Element Heat Transfer Code", University of California, Lawrence Livermore National Laboratory, Rept UCID-20484, 1985.
- 3 J. O. Hallquist, "NIKE2D - A Vectorized Implicit, Finite Deformation, Finite Element Code for Analyzing the Static and Dynamic Response of 2-D Solids with Interactive Rezoning and Graphics", University of California, Lawrence Livermore National Laboratory, Rept UCID-19677, 1986.
- 4 J. O. Hallquist, "NIKE3D - An Implicit, Finite Deformation, Finite Element Code for Analyzing the Static and Dynamic Response of 3-D Solids", University of California, Lawrence Livermore National Laboratory, Rept UCID-18822, 1984.
- 5 R. L. Thompson and R. L. Maffeo, "A Computer analysis Program for Interfacing Thermal and Structural Codes", NASA Lewis Research Center, Rept NASA-TM-87021, 1985.
- 6 R. Maffeo, "TRANCITS Program User's Manual", General Electric Company, Cincinnati, OH, NASA Rept CR-174891, 1985.
- 7 R. Maffeo, "Burner Liner Thermal/Structural Load Modeling", General Electric Company, Cincinnati, OH, NASA Rept CR-174892, 1984.
- 8 A. B. Shapiro, "FACET - A Radiation View Factor Computer Code for Axisymmetric, 2D Planar, and 3D Geometries with Shadowing", University of California, Lawrence Livermore National Laboratory, Rept UCID-19887, 1983.
- 9 T. J. R. Hughes, The Finite Element Method, Prentice Hall, 1987.
- 10 J. O. Hallquist, "ORION - An Interactive Color Post processor for 2-Dimensional Finite Element Codes", University of California, Lawrence Livermore National Laboratory, Rept UCID-19310, 1985.

END

DATE FILMED

10 / 25 / 90