ANL-00/33

User Guide for Modeling of Electroconsolidation Process with ELEKTRA and MaPS Codes

by F. C. Chang, H. M. Domanus, and R. A. Valentin

Argonne National Laboratory, Argonne, Illinois 60439
operated by The University of Chicago
for the United States Department of Energy under Contract W-31-109-Eng-38

Energy Technology Division
Energy Technology Division
Energy Technology Division
Energy Technology Division
Energy Technology Division
Energy Technology Division
Energy Technology Division
Energy Technology Division

Argonne National Laboratory, with facilities in the states of Illinois and Idaho, is owned by the United States Government and operated by The University of Chicago under the provisions of a contract with the Department of Energy.

Available electronically at http://www.doe.gov/bridge

Available for a processing fee to U.S. Department of Energy and its contractors, in paper, from:

U.S. Department of Energy
Office of Scientific and Technical Information
P.O. Box 62
Oak Ridge, TN 37831-0062
phone: (865) 576-8401
fax: (865) 576-5728
email: reports@adonis.osti.gov

ARGONNE NATIONAL LABORATORY
9700 South Cass Avenue, Argonne, Illinois 60439

ANL-00/33

User Guide for Modeling of Electroconsolidation Process with ELEKTRA and MaPS Codes

by
F. C. Chang, H. M. Domanus, and R. A. Valentin
Energy Technology Division

January 2001

Work sponsored by
U.S. Department of Commerce
NIST Advanced Technology Program

Contents

Figures

Tables

User Guide for Modeling of Electroconsolidation Process with ELEKTRA and MaPS Codes

F. C. Chang, H. M. Domanus, and R. A. Valentin

Abstract

Electroconsolidation® is a proprietary process for rapid, pressure-assisted densification of powder preforms. The parts to be consolidated are placed within a bed of free-flowing powder, and pressure is applied by rams that act on the powder medium. Heat for consolidation is generated internally by electrical resistive heating of the medium while it is in direct contact with the preform(s).

A mathematical model has been developed to predict the temperature profiles within the Electroconsolidation die during operation. Numerical simulation is performed by coupling a three-dimensional (3-D) finite-element electromagnetic field program (ELEKTRA) and a 3-D finite-difference materials process simulator (MaPS) to incorporate the effects of pressure and electrical fields on heat transfer and temperature variation within the die during heating and cooling.

The methods used in developing the model are presented. Use of the model minimizes the experiments needed to design and evaluate commercial applications of Electroconsolidation.

1 Introduction

Electroconsolidation® is a proprietary process for rapid pressure-assisted densification of preformed materials by the simultaneous application of high pressure and ultra-high heat [1,2]. It can be used to manufacture parts from metal, ceramic, and polymer powder preforms. The process offers advantages when pressure and heat must be applied simultaneously to the part to be consolidated. Preformed materials of complex shape can be consolidated directly to near-net-shape articles by Electroconsolidation, which is referred to as a "soft-tooling" or "pseudo-fluid" process that utilizes a particulate solid as the pressure-transmitting medium (in the case under discussion, graphite). It differs from other processes of this type in that the preform is heated by electrical resistance of the pressure-transmitting medium while the medium is in contact with the preform inside the die chamber.

Electroconsolidation offers advantages over the established methods of pressure-assisted consolidation such as hot pressing (HP) and hot isostatic pressing (HIP). A major advantage is the ability to consolidate parts of complex shape to near-net shape without the need to clad the part. It offers the opportunity to use very high temperatures (above 2500°C if needed). Cycle times can be very short because heating takes place in the immediate vicinantage is the apiece. The heating rate can be extremely rapid. Electroconsolidation can reduce the cycle times of several hours needed in HP or HIP. Construction of the apparatus is simple and the operation of the process offers considerable flexibility. Electroconsolidation can produce components more cost-effectively than HP, HIP, or other consolidation methods. It is especially suited to making large numbers of small identical parts. Equipmentncosts can be substantially lower for Electroconsolidation than for HIP, and existing pressing equipment can be retrofitted for Electroconsolidation.

The basic Electroconsolidation apparatus is shown schematically in Fig. 1. The part (preform) surrounded by the pressure-transmitting graphite medium is contained within a cylindrical chamber (the die). Pressure is applied uniaxially by rams that enter the die from both top and bottom. The rams are low-resistance electrical conductors connected to a power source to facilitate the passage of current through the pressure-transmitting medium, which is also electrically conductive. Heating to the consolidation temperature is achieved by passing an electric current through the medium, thus causing it to be heated electrothermally. Heat is transferred from the medium to the preform by conduction concurrently with the application of compaction pressure.

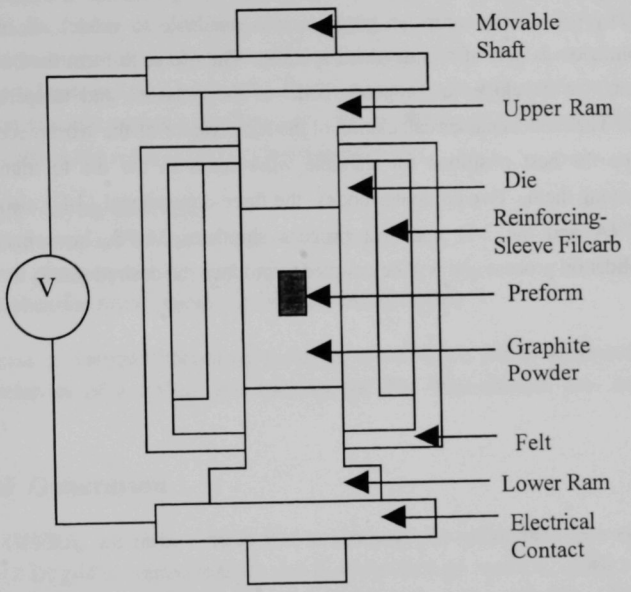


Fig. 1. Schematic diagram of Electroconsolidation apparatus

The ideal medium should be free-flowing so it fills all voids and compresses uniformly against the contour of the part; it should be chemically inert and stable at high temperature; and it should be electrically conductive, yet offer adequate electrical resistivity to act as the resistive heating source in the system. It should also be resiliently compressible so it can be compacted at high pressure without breaking, and readily released from the consolidated part when the pressure is reduced. A spherical form of porous graphitic carbon (75-500 μm) is close to the ideal medium for this process.

Because the electrical properties of the medium vary with pressure and temperature, temperature within the die chamber may vary and lead to over- or under-heating and distortion of the part. Any direct measurement of pressure or temperature within the die would require insertion of sensors with external leads and would be impractical in production operations, especially when internal temperatures typically may be 2000-3000°C. Electroconsolidation is a process that involves interaction among electrical, thermal, mechanical, and powder metallurgical phenomena [3-6]. Therefore, to establish the control system and commercialize this technology, a computer

model is needed to predict temperatures at various locations in the die as a function of time and to minimize expensive, time-consuming full-scale testing.

Previously, no single computer program was available to model all of the features of Electroconsolidation as part of an integrated system. Therefore, to form the basis for a process-control system, we developed a computer model of the pressure and temperature distributions inside the die. The model enables calculation of the temperature of the workpiece. The model can help determine the best positions for multiple workpieces in the die to minimize temperature differences among them. Two computer codes, the three-dimensional (3-D) electromagnetic (EM) field ELEKTRA and the 3-D materials process simulator MaPS, have been used with the Electroconsolidation process and will be coupled to produce the desired result.

2 Synopsis of ELEKTRA Code

The commercial software ELEKTRA was developed by Vector Fields Inc. in England for 3–D computation of eddy currents and electromagnetic design. The ELEKTRA package is a sound-generation-integrated suite of programs for 3-D analysis of eddy currents. The package includes powerful pre- and post-processing modules linked to an advanced-design analysis module that is based on research carried out at Vector Fields and Philips, Eindhoven. The package is used in many applications and is specifically characterized by

- 3-D time-varying EM fields,
- Time harmonic and transient fields and motion effects,
- Efficient geometry and data input facilities,
- Comprehensive result processing and 3-D model display.

ELEKTRA uses a discrete finite-element model to solve the partial differential equations that govern the behavior of a system and includes the EM finite-element pre- and post-processor OPERA-3D.

2.1 Model Generation

Using OPERA, we form a mesh that is automatically subdivided into elements. A two-dimensional (2-D) grid is created initially and is swept through space to create a 3-D model. The sweep operation includes facilities for rotation, projection, and translation. The resultant model has the following features:

- 3-D curved surfaces
- Tetrahedral, prism, and brick primitives
- Separately modeled conductors
- Wire frame and hidden surface displays
- Light-source shading option for display purposes.

The mesh primitive blocks are assigned orientation and material characteristics. The resultant input is fed directly into the analysis module.

2.2 Analysis

ELEKTRA solves the 3-D time-varying magnetic field equations. The time variation can either be transient or alternating (steady-state). In addition, the effects due to motion can be computed. Total and reduced magnetic scalar potentials are used in nonconducting media to reduce the solution costs and correct the cancellation errors associated with the simple reduced-potential approach. In conducting media, the program uses the magnetic-vector and electric-scalar potentials, which are directly coupled with the potentials on the exterior. The use of magnetic vector and electric scalar potentials enables accurate modeling of both solutions and differing conductors in contact.

2.3 Post-Processing

OPERA provides the user facilities to display the results of the analysis in several ways, including

- 3-D model views from any angle
- Graphs, histograms, and contour maps of the solutions
- Contours of components of the results on any surface
- Calculation of fields, forces, and energy
- Particle tracking
- User-defined function.

3 Synopsis of MaPS Code

Argonne National Laboratory (ANL) and Superior Graphite Company have worked together under the National Institute of Standards and Technology (NIST) Advanced Technology Program (ATP) to modify ANL software so it could help the powder industry model the Electroconsolidation process at substantial production and energy cost savings. This joint effort has provided a 3-D computer model, the macroscopic materials process simulator (MaPS), that can predict pressure and temperature profiles in the Electroconsolidation die chamber. The MaPS model is verified by predicting pressure and temperature in the die chamber for application at an Electroconsolidation setup at ANL.

MaPS significantly shortens the time required for powder consolidation research that involves the use of EM fields for electrical resistivity heating. The model also optimizes existing Electroconsolidation processes and minimizes expensive, time-consuming, full-scale testing. The cost savings from Electroconsolidation would give the powder products an enormous economic advantage over products produced by competing methods.

The above-described work was supported by the Department of Commerce (DOC), NIST ATP. ANL's work under the ATP is to develop a computer model for the Electroconsolidation process for powder consolidation.

3.1 Features of MaPS Code

The MaPS software combines heat transfer, fluid flow, and EM field aspects, and can be used to describe various consolidation aspects. MaPS is a 3-D time-dependent computer code that uses a finite-volume formulation for the mass, momentum, energy, and turbulent-flow equations; its origin is in CaPS [7,8] and it exhibits the following characteristics:

- MaPS uses the Casting Pre-processor (CPRE) geometric modeling package to construct the geometry, generate a neutral file that consists of a list of named components, and post-process the simulation results; it builds the geometry independently of the mesh, a timesaving procedure. Each named component involves a set of hyperpatches, patches, or grids that are included in the session file to create the geometry.

- A mesh generator of structured regular cells is included and is interfaced with the neutral-file output of the solid geometric package.

- Visual user interfaces that are based on the HOOPS package, which contains a hierarchical database of geometric information, have been developed. These visual interfaces allow the user to observe, create, verify, and view the meshed geometry,

set up the boundary and initial conditions, view the various named components, and initialize the simulation parameters (e.g., maximum time allowed for the run, iteration time interval, maximum number of steps, step interval for graphical output, etc.).

- The MaPS generates a data file to read EM-field data from ELEKTRA into MaPS.
- The MaPS shell scripts interactively provide a step-by-step procedure to simulate the solidification process, thus making the software very user friendly.

MaPS efficiently solves transient heat conduction within the metal and between the metal and the mold and should be used in conjunction with CPRE and HOOPS packages. With MaPS, containment techniques can be improved by validating the design of the edge dam system to increase both magnetic field and magnetic force. Also, optimization can be achieved by computing the effects of key parameters in the process. MaPS is an efficient software tool that saves time and energy and reduces surfacing work while producing powder products for Electroconsolidation.

3.2 Organization of MaPS Model

An overview of the MaPS is provided to inform the user about what this software requires and what it can do. A flow chart is provided in Section 3.4 of this report to show the various modules of the MaPS.

The first step in modeling with MaPS software is to build the geometry. MaPS interacts with CPRE via the neutral file. The step after building the geometry is to either divide the geometry into rectangular cells (meshing) or set the boundary and initial conditions so the various configurations are similar to those used in real procedures. Once the geometry is set and the boundary and initial conditions are assigned to various parts of the geometry, geometric meshing can be performed.

The input file to begin the simulation is described in Sec. 11.1. Section 11.2 describes how hard-copy output can be obtained for various parameters with the Print Step feature of MaPS. In MaPS, material properties are temperature-dependent and are to be included in the properties database directory.

As mentioned earlier, MaPS software originated from the CaPS code and is extended to magneto-hydrodynamics (MHD) application by accounting for the EM-field effect. For more information and guidelines, the reader is referred to Refs. 9 and 10.

3.3 Overview of MaPS Modules

Several modules of MaPS can be used to facilitate the geometric setup and to submit a simulation run. The MaPS shell scripts perform all of the linkages, and the user generally need not worry about file structure.

The MaPS shell scripts make up an important user-friendly tool that links all files and directories and thus does not burden the user with creating links, etc. The visual interfaces (Vbounds and Vmesher) facilitate the setup of initial and boundary conditions and geometry meshing. Post-processing can be performed either with the Print Step load module that provides hard-copy output of the variables for particular time steps, or with casting post-processor (CPOST), which is linked to the Cas load module via Capspat. Thus, the simulation results can be displayed in CPOST, and the various plots of velocity vectors, solid mass fraction, temperature isotherms, and fluid volume can be analyzed to understand fluid flow and heat transfer for the part assembly in the geometry.

Next, we will discuss how various input and output files are linked and placed in the working directory. MaPS creates the files according to the working modules and links or copies the various files in the current working module. In this way, the time lost in input/output processes is minimized.

3.4 Flow Chart of MaPS Software

Figure 2 shows the flow chart of the MaPS software and indicates the modules that must be set and initialized before going to the next module down the flow chart. Initially, the geometry is set up with the CPRE software.

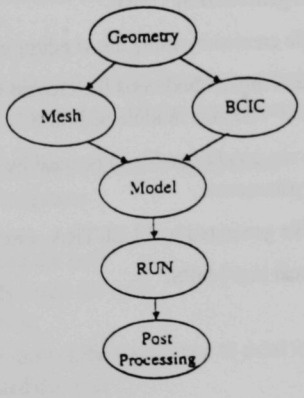


Fig. 2. Flow chart of MaPS software

The geometry is built by using grids, patches, and hyperpatches. Patches and hyperpatches are grouped into user-defined, named components to facilitate boundary condition and initial condition (BCIC) assignments, respectively. These named components are the basis upon which the boundary conditions identify various sections of the geometry. Next, a neutral file is created.

Because this neutral file contains the descriptions of the named components, it is a link that enables MaPS to compute the geometric hyperpatch volumes, display the geometry in the visual interfaces, and identify the various geometric sections.

The meshing of the geometry and the setup of the initial and boundary conditions are on the same level and thus either module can be initiated. The geometric volumes created with CPRE can be initialized and the surfaces can be assigned specific boundary conditions with the Vbound module. The meshing is performed by the visual Vmesher interface. These visual interfaces can easily detect any geometric flaws. If flaws are detected, the user must start again by creating or rectifying the geometry with CPRE and then, after computing the hyperpatch volumes, initiate the Vbounds and Vmesher again.

After the user sets the problem, meshes the geometry, and applies the boundary and initial conditions, a model is created with one particular mesh and one particular set of conditions. The model includes the various conditions under which the simulation runs are to be conducted. Multiple runs can be carried out for the same model by changing the conditions of the simulation via the *.TS file.

Once the runs are completed, the post-processing module can be initiated and the simulation results may be viewed. The files that show up in the simulation are

*NE	Neutral file generated by CPRE.
*ME	Meshing file created by using the vi editor or the Vmesher module of MaPS.
*BC	Boundary and initial conditions file created with the vi editor of the Unix system or the Vbounds module of MaPS.
*TS	Simulation run conditions file to be used by MaPS to set the various simulation parameters.
*EMF	EM fields file generated by ELEKTRA, which is used to enter the electrical resistivity heat into MaPS.

4 Density Modulus and Program

In the Electroconsolidation model, an independent program R-C-CVP.f is developed to calculate the distributions of density, electrical resistivity, and thermal conductivty of the particulate medium after compaction. This program is used for both single- and double-action pressing, and is also good for both simple and preliminary (Coulomb's law) pressure models. Electrical resistivity output data are assigned to each compacted region of the medium bed in ELEKTRA. Density and thermal conductivity data are also used for each compacted region of particulate medium in MaPS. The sample program and the key parameters are listed below for your understanding and reference:

```
c -------------------------------------------------------------------------------------------------
c -------------------------------------------------------------------------------------------------
c   Program Name (R-C-CVP.f)
c   Calculations of Density, Electrical Resistivity, and Thermal Conductivity
c   for Electroconsolidation Process
c
c   Single/Double-Action Pressing
c   Simple/Preliminary (Coulomb's Law) Pressure Model
c
c   item1     : number of pressure-transmitting medium regions
c   item2     : number of temperature set for thermal conductivity
c
c   icase     = 0  simple pressure model
c   icase     = 1  Coulomb's law pressure model for single-action
c   icase     = 2  Coulomb's law pressure model for double-action
c
c   dens0     : density of particulate medium before compaction
c   pram0     : applied pressure from rams
c   compjc    : Janssen constant
c             = 0.204 for lubricated die wall
c             = 0.175 for non-lubricated die wall
c   dofmed    : inside diameter of die
c   findex    : friction index (the ratio of the shear force to axial force)
              = 0.055 for lubricated die wall
c             = 0.11   for non-lubricated die wall
c   hofmed    : height of particualte medium bed
c
c   zz        : Z coordinate of each layer in Coulomb's law pressure model
c   pax       : axial pressure of Coulomb's law pressure model
c   pra       : raidal pressure of Coulomb's law pressure model
c   dens      : density of particulate medium after compaction
```

```
c    temp       : temperature set used for thermal conductivity
c    condk      : thermal conductivity of particulate graphite medium
c    covol      : compacted volume, % of initial volume
c    econd      : electrical conductivity
c    press      : applied pressure in the particulate medium
c    resis      : electrical resistivity
c    condii     : thermal conductivity of solid graphite medium
c ------------------------------------------------------------------------------------------------------
c
     parameter (item1=2,item2=6,item3=11)
     dimension covol(item1),resis(item1),econd(item1),
    1          dens(item1),press(item1),temp(item2),
    2          condii(item2),pax(item3),pra(item3),
    3          condk(item1,item2),zz(item3)
      open (6,file='OUTPUT')

     data icase/0/
      data dens0,pram0 /1040.0,5000.0/
      data compjc,findex,hofmed,dofmed/0.175,0.11,0.1143,0.127/
     data covol/70.0,64.5/
      data temp /25.0,260.0,538.0,816.0,1093.0,1649.0/
      data condii/173.1,112.5,83.1,60.6,47.6,31.2/
     data zz /-0.033782, -0.023782, -0.013782,      -0.01,     -0.005,
    1                0.0,       0.005,         0.01, 0.013782, 0.023782,
    2          0.033782/
c
     coeff  = 2.0*findex/dofmed

     if(icase.eq.1) then

       do k=1,item1
         zavg     = 0.5*(zz(k)+zz(k+1))
         zc       = 0.5*hofmed-zavg

         pax(k)   = pram0*exp(-2.0*coeff*zc)

c        pax(k)  = pram0*exp(coeff*(2.0*zavg-hofmed))
         pra(k)   = compjc*pax(k)

         write(6,10) k,zavg,pax(k),pra(k)
   10      format(/,'k =',i3,3x,'zavg =',f8.5,' m',3x,
     1           'pax =',f7.1,' psi',3x,
     2           'pra =',f7.1,' psi')
       end do

     elseif(icase.eq.2) then
```

```
      do k=1,item1
        zavg    =0.5*(zz(k)+zz(k+1))
        if(zavg.ge.0.0) then
        pax(k)  = pram0*exp(coeff*(2.0*zavg-hofmed))
        pra(k)  = compjc*pax(k)
        else
        pax(k)  = pram0*exp(-coeff*(2.0*zavg+hofmed))
        pra(k)  = compjc*pax(k)
        endif
        write(6,20) k,zavg,pax(k),pra(k)
20      format(/,'k =',i3,3x,'zavg =',f8.5,' m',3x,
     1          'pax =',f7.1,' psi',3x,
     2          'pra =',f7.1,' psi')
        end do

      endif

      if(icase.eq.0) then

      do k=1,item1
        x       = covol(k)
        dens (k)= 100.0*dens0/x
        press(k)  = 1.2873e+5 - 4015.1*(x) + 43.861*(x**2)
     1          - 0.16764*(x**3)
        p       = press(k)
        if(p.gt.10000.0) p = 10000.00d00
        resis(k)  = 2.14e-2 - 9.0154e-6*(p) + 1.9851e-9*(p**2)
     1          - 2.0105e-13*(p**3) + 7.5175e-18*(p**4)
        econd(k) = 1.0/resis(k)
        write(6,30) k,covol(k),dens(k),press(k),resis(k),econd(k)
30      format(//,'item1    =',i5,3x,/,
     1          'volume    =',f7.2,' % of initial volume',/,
     1          'density   =',f7.1,' kg/m**3',/,
     1          'pressure  =',f9.2,' psi',/,
     1          'resistivity =',f7.4,' ohm-cm',/,
     1          'electrical conductivity =',f7.2,' Siemens/cm',/)

        do ij=1,item2
          t     = temp(ij)
          ratio = condii(ij)/condii(1)
          condk(k,ij) = 418.4*0.00031*ratio/resis(k)
          write(6,40) temp(ij),condk(k,ij)
```

```
 40     format(
     1              'T =',f8.1,' C',5x,
     1              'k =',f7.2, ' watt/(m-C)')
       end do
     end do

     elseif(icase.eq.1.or.icase.eq.2) then

     do k=1,item1
       p           = 2.0*pax(k)+4.0*pra(k)
       resis(k)    = 0.001 * (15.38 - 2.022e-3*(p) + 1.221e-7*(p**2)
     1             - 2.474e-12*(p**3))
       econd(k)    = 1.0/resis(k)
       covol(k)    = 99.622 - 5.4352e-3*p + 2.5299e-7*p**2
     2             - 4.842e-12*p**3
       dens(k)     = 100.0*dens0/covol(k)
       write(6,50) k,covol(k),dens(k),p,resis(k),econd(k)
    50 format(//,'item1       =',i5,3x,/,
     1          'volume      =',f7.2,' % of initial volume',/,
     1          'density     =',f7.1,' kg/m**3',/,
     1          'pressure    =',f9.2,' psi',/,
     1          'resistivity =',f7.4,' ohm-cm',/,
     1         'electrical conductivity =',f7.2,' Siemens/cm',/)

       do ij=1,item2
         t      = temp(ij)
         ratio  = condii(ij)/condii(1)
         condk(k,ij) = 418.4*0.00031*ratio/resis(k)
         write(6,60) temp(ij),condk(k,ij)
    60   format(
     1             'T =',f8.1,' C',5x,
     1             'k =',f7.2, 'watt/(m-C)')
       end do
     end do

     endif
    stop
    end
c ------------------------------------------------------------------------------
c ------------------------------------------------------------------------------
```

After entering the required data in the R-C-CVP.f program, the user must compile and run the program:

% f77 R-C-CVP.f (use Fortran complier to compile program)

% a.out (run the calculations of the program)

An output file OUTPUT will be automatically produced to provide the required density, electrical conductivity, and thermal conductivity properties of the particulate medium under differing compactions. The following is a sample OUTPUT file.

```
-----------------------------------------------------
item1 =  1
volume =  70.00 % of initial volume
density = 1485.7 kg/m**3
pressure =  5091.39 psi
resistivity =  .0055 ohm-cm
electrical conductivity = 182.68 Siemens/cm

T =   25.0 C    k =  23.69 watt/(m-C)
T =  260.0 C    k =  15.40 watt/(m-C)
T =  538.0 C    k =  11.37 watt/(m-C)
T =  816.0 C    k =   8.29 watt/(m-C)
T = 1093.0 C    k =   6.52 watt/(m-C)
T = 1649.0 C    k =   4.27 watt/(m-C)

-----------------------------------------------------
item1 =  2
volume =  64.50 % of initial volume
density = 1612.4 kg/m**3
pressure =  7244.90 psi
resistivity =  .0045 ohm-cm
electrical conductivity = 220.44 Siemens/cm

T =   25.0 C    k =  28.59 watt/(m-C)
T =  260.0 C    k =  18.58 watt/(m-C)
T =  538.0 C    k =  13.73 watt/(m-C)
T =  816.0 C    k =  10.01 watt/(m-C)
T = 1093.0 C    k =   7.86 watt/(m-C)
T = 1649.0 C    k =   5.15 watt/(m-C)
-----------------------------------------------------
```

5 Model Setup of ELEKTRA [9,10]

5.1 Starting Software

The program can be started by typing

Opera

If necessary, select "3D" analysis (if the 2D software is also available on the system, this response will be needed). The program will display its start-up menu. At the beginning, simply select the pre-processor:

pre

5.2 Menu System

If menus are available, the software can be controlled by either a menu system, or by keyboard input. The menus are activated by using the cursor and clicking the left mouse button. The primary level menu items are

FILE	to read and write data and command files, and ending the program.
OPTIONS	to control colors, graphical output, titling, etc.
DISPLAY	to control the dimensions and style of the picture of the model.
HELP	to get help on how to use the program.
DEFINE	to define model data.
MODIFY	to change model data.
MENU_OFF	to switch to keyboard command mode. The menu system can be turned on again by typing ^<return> (the caret character ^ followed by a carriage return).

5.3 Menu Mode Script

To indicate the use of the mouse for graphical input in this menu mode script, the coordinates (approximate) at which the cursor should be positioned are given within angled brackets < > for example, <0,0> mouse.

5.3.1 Finite-Element Mesh

In the following instructions, sequences of menu selections will be presented in the manner shown below.

"DEFINE"

DEFINE is the command that puts the OPERA-3d pre-processor into the mode needed to create the finite-element mesh. The preprocessor only supports one type of finite-element-mesh construction, i.e., a surface defined by a set of facets is defined and is then extruded or swept through space to create a volume discretisation. The user input is structured by the program. Points used to define the geometry are defined first, followed by the surface facets and the element subdivision of these facets. It is possible to define more points while facets are being defined, and more facets while subdivisions are being defined. The surface of facets is then extruded or swept through space, thus creating a series of volume layers. The topology of the initial surface is maintained during the extrusion operations, but the point coordinates can be changed in the new surface created by each extrusion or sweep. Once the complete volume model has been created, the volumes in the layers of the mesh can be assigned material attributes and boundary conditions can be assigned to the faces of the volumes.

The next parameter on the DEFINE command is the type of mesh that allows definition of the finite-element mesh for the solution of the problem. In Electroconsolidation, the option "Finite element mesh" is used to analyze modeling. The finite element mesh can be modified by using "MODIFY," and have extra layers added by using "EXTEND." The steps of DEFINE process are

Define new mesh →	Finite-element mesh →	XY plane, extrude in Z.
	8 or 20 node conductors	

These steps indicate that each of the items shown should be selected in sequence. The last menu shown is replaced by a sequence of menus and input boxes that lead through the definition process; the first is a parameter box, which requests the value of the W coordinate of the base plane. Enter the value by typing on the keyboard.

W coordinate of plane = 0 or the coordinate of base plane

The next parameter box requests the coordinate limits for the graphics window. The values should be entered as follows:

Minimum on horizontal axis	=
Maximum on horizontal axis	=
Minimum on vertical axis	=
Maximum on vertical axis	=
ACCEPT	DISMISS

Fill in the parameter box for each line in turn. When a parameter name (e.g. Minimum on horizontal axis) is highlighted, its value can be edited. If the value is highlighted, typing a new

value will replace the old. Having filled in the parameter box for the window coordinate limits, select ACCEPT with the mouse or, if ACCEPT is highlighted, hit Return to issue the command.

Labels are assigned to each entity (i.e., point, line, facet, or volume) in the mesh. All entities have the label ALL; facets have boundary condition names; and volumes have material names, potential types, and element types. Additional labels can be added or removed with the "LABEL" command. These labels can be used to select subsets of the entities for "DISPLAY".

Each time the DEFINE command is used to create a finite-element mesh, a new mesh is started. Up to 100 meshes can be defined; each one forming part of any data set that is prepared for analysis. The user is responsible for ensuring that the meshes are contiguous at their interfaces.

5.3.2 Points on the Base Plane

The user must specify points to define the corners and midside points of the facets in the base plane. Additional points may be entered by returning to Point Definition Mode from Facet Definition Mode. It is also possible to copy points and facets in the Facet Group Operations Mode.

Points are defined by using the graphics cursor or the keyboard. The cursor can be used to position points at the crosshair position, or at points on construction lines. Keyboard input can be in Cartesian or cylindrical polar coordinates with respect to the origin of the current local coordinate system. When enough points have been defined, leave the Point Definition Mode and move on to Facet Definition Mode.

A dialog box appears when the Point Definition menu is initiated. The most appropriate options are "Give U, V, W" or "Give R, Theta, W" which allow the user to specify the Cartesian or Polar coordinate of each point. This is done by means of a dialog box:

Cartesian Coordinate Input			Polar	
U coordinate			R coordinate	
V coordinate		or	T coordinate	
W coordinate			W coordinate	
Accept	Exit		Accept	Exit

When the values are correct, selecting ACCEPT defines the point and an X should appear at the correct location on the graphics window. The other points can now be entered in the same way. After the last point has been entered and accepted, close the dialog box with "Exit" and "Go to Facet Definition."

5.3.3 Facets on the Base Plane

Facets are defined by connecting the points on the base plane; they can be triangles or quadrilaterals with straight or curved edges. Points are selected in sequence as corners (with menu item "Corner") or midpoint (with menu item "Mid-side"). For triangles or quadrilaterals with no midpoint on the fourth side, menu item "Close" can be used to close the facet.

Sides with midside points are quadratic. The midside points must be between 1/4 and 3/4 points along the length of the side (this is verified by the "CHECK" command). If the midside point is not half way along the side, the discretisation will also vary quadratically, with smaller elements near the corner that is closer to the midside point. This can be used to grade the mesh, even for straight-sided facets.

Points that have been used as corners cannot subsequently be used as midside points, and vice-versa. Additional points may be defined by returning to Point Definition Modes (menu item "Return to points"). Facets can be copied and transformed by using the Facet Group Operations Mode, which can be entered by using menu item "Go to Group Ops."

Care must be taken to ensure that the entire base plane is covered with facets. This is especially so if the available graphics facilities do not shade faces as they are defined. When the base plane is complete, menu item "Go to Subdivisions" leaves the Facet Definition Mode and moves on to the Base Plane Subdivision Mode. It is possible to return to the Facet Definition Mode from the Base Plane Subdivision Mode to define more facets.

5.3.4 Element Subdivision of Base Plane Facets

The volumes created by extruding the initial surface plane (base plane) are treated as super-elements. They are normally divided into smaller elements that are used for the actual finite element approximation. The division of the volume into elements is determined by the subdivision defined for the facet edges and the subdivision specified for each layer of mesh created by an extrusion operation.

A default subdivision of 1 is initially set for all facet edges. When all the edges have been updated, the program will reply that the subdivision is complete. Only regular subdivision is supported by analysis programs; opposite faces of the volumes must have equal numbers of divisions. The subdivision defined for an edge is therefore carried through and displayed for all adjoining facets.

Triangular facets are meshed by mapping onto a quadrilateral with two corners coincident. Therefore, triangles must have two sides with the same number of subdivisions. It is not possible to leave the subdivision phase until this condition is satisfied. The subdivision of each edge is uniform, unless the edge is a quadratic line with the midpoint not at the geometric midpoint of the

line. In that case the elements at the end of the edge closer to the midpoint will be smaller than those at the other end. If the subdivision is given for a single edge and that edge is one side of a quadrilateral facet, then the subdivision of the opposite edge will be set to the same value. The subdivision will be carried through the mesh until a triangular facet, the edge of the mesh, or a facet already set in this operation is reached.

The Subdivisions menu appears. In menu mode, the "Set subdivisions" option starts a sequence of two operations:

• If the number of elements required is less than 10, the number can be selected from the buttons in the dialogue box. Otherwise, any value can be entered into the "Other" input box.

Subdivision
1 2 3
4 5 6
7 8 9
Other
Accept

• The subdivision can be applied to individual lines or to all lines. If the subdivision is applied to individual lines, the program copies the subdivision to the opposite sides of four-sided facets. Sides are selected by pointing inside a facet that is close to the required side.

The full sequence of operations is as follows:

Set subdivision
Select button 4 and Accept
Apply to line
Return

Set subdivision
Select button 3 and Accept
Apply to line
Return

Set subdivision
Select button 5 and Accept
Apply to line
Return

The program responds "The in-plane subdivision is complete." Press any key on the keyboard or click a mouse button to dismiss the message and then "Go to Extrusions."

5.3.5 Extrusions in the Third Dimension

Once the initial surface set of facets has been defined and subdivisions have been assigned to the edges, the program moves on to apply extrusion operations to the set of facets. At least one extrusion operation must be applied to the set of facets, but many others may be needed to define the complete problem. It is also possible to add more extrusions to a complete mesh by using the EXTEND command. The EXTEND command must be used to add the second and subsequent extrusions if the data are defined in menu mode.

In menu mode, a LINEAR or QUADRATIC extrusion can be chosen from a menu. The dialog box that follows allows the user to give the coordinate of the next plane and the number of subdivisions, and to select GLOBAL or RELATIVE. The values and options are interpreted as follows:

• Coordinate of next plane. The W-coordinate can be interpreted as GLOBAL or RELATIVE. When a GLOBAL coordinate is given for the new plane, the extrusion is formed by projecting the current facet set in the W direction until the W-directed lines intersect with the plane defined by the global value of W. The W-coordinate can also be interpreted as a RELATIVE move in the W direction, in which case the coordinate value is added to the W-coordinates of the points in the current plane.

• Subdivision. This specifies the number of layers of elements there will be between the two planes. The subdivision is ignored when the option DEFINE is being used for conductor elements.

• Linear or Quadratic. The W-directed lines can be LINEAR or QUADRATIC. Initially, the lines are created linear, i.e., straight; but if QUADRATIC is selected, a midextrusion plane of points is also created, at the geometric midpoints of the lines. This means that QUADRATIC lines can be changed to be curved, or to have nonuniform subdivision by moving the points on the mid-extrusion plane away from the geometric midpoints of the lines.

In Electroconsolidation analysis, the Extrusion definition menu appears. Select a Linear extrusion. This brings up a dialog box, into which the extrusion parameters should be entered.

Extrusion Definition
Coordinate
Global Relative
Number of elements
Accept

When the extrusion data are complete, select Accept. The program responds "Layer 1 created." Press any key on the keyboard or click a mouse button to dismiss the message. No point editing is needed so select Finish Editing in the Point Editing menu.

5.3.6 Material Properties

The extrusion operations create a discretisation of space. It is now necessary to set the material and mesh properties within each volume. The program presents the user with each layer in turn and the user sets values, which override the default or current setting within each volume.

In menu mode, the menu item "Select/de-select volume" allows volumes to be added or removed from a list. Menu item "Select and define" adds one last volume to the list and causes the program to display a dialog box into which the material properties can be entered. Other menu items allow the properties to be summarized in all of the volumes ("Show") or to be listed ("List") for an individual volume.

The first compulsory keyword is the material name. This can be (almost) any string of up to eight characters, beginning with a letter. Two material names are predefined, "AIR" and "NULL." AIR is for any nonconducting volume with a relative permeability of unity. Volumes with name NULL are omitted from the final mesh, enabling the creation of holes in the mesh to represent e.g., electrodes. The definition of each material in terms of its permeability and, if necessary, conductivity is supplied during the commands that create the analysis data files.

The second compulsory keywords refer to the potential type. These keywords are "REDUCED," "TOTAL," and "VECTOR." The REDUCED scalar potential can be used for any material and must be applied to volumes that will contain source conductors. It should only be used in non-AIR materials if the source conductor and material geometry would make a total scalar potential volume multiply connected, or source currents flow in the iron. The TOTAL scalar potential should be used for electrostatics problems and magnetics problems. It can also be used for AIR and any nonconducting and nonlinear permeable material. The VECTOR potential is for conducting materials that carry currents in time-dependent analysis.

Optional keywords that define properties are LINEAR and QUADRATIC and define the element types to be used. LINEAR elements are 8-noded and QUADRATIC elements are 20-noded isoparametric. Optional keywords for setting additional volumes with the same definition are "ALL," "FROM," and "TO." ALL indicates that all of the volumes in the layer(s) should be set the same. Both FROM and TO should be followed by numeric values that specify a number to specify a range of layers that are to be set the same. If either is omitted, the current layer number is assumed. The numeric value after TO can be specified as "*" to indicate the top layer. The material names, potential types, and element types are stored as labels on each volume and can be used to select parts of the model in the DISPLAY command.

In Electroconsolidation, the users use LINEAR element types for all materials and use potential types as follows:

- Preform (VECTOR potential)
- Particulate Medium Bed (VECTOR potential)
- Ram (VECTOR potential)
- Die (VECTOR potential)
- Reinforced-Sleeve Filcarb (VECTOR potential)
- Insulation Felt (VECTOR potential)
- Electrical Insulation Coating (VECTOR potential)
- Electrical Insulation Silcon Carbide (VECTOR potential)
- Surrounding Air (REDUCED potential)

5.3.7 Boundary Conditions

Boundary conditions can be set on any surface of any volume in the mesh. To achieve this, the program presents the facets in separate groups. In menu mode, three groups of facets are present: first the facets on the base plane; second, the top plane of the mesh; and third, the extrusion facets normal to the planes. The menu item "Select/de-select facet" allows facets to be added or removed from a list. Menu item "Select and define" adds one last facet to the list and causes the program to display a dialog box into which the boundary conditions can be entered. The menu item "List conditions" allows the boundary conditions to be listed for an individual facet.

Boundary conditions can be imposed in two ways: (1) by restricting the "MAGNETIC" or "ELECTRIC" field to be "NORMAL" or "TANGENTIAL," which sets combinations of the potentials and derivatives, as appropriate, for the type of problem and volume concerned; and (2) by setting values of potential or the normal derivative of scalar potential or components of vector potential. If necessary, a potential boundary condition can be assigned in addition to the normal or tangential conditions to override the zero values of the scalar potential.

Optional keywords for setting additional facets with the same definition are "ALL," "FROM," and "TO." ALL indicates that all the facets in the plane(s) or layer(s) should be set the same. Both FROM and TO should be followed by a numeric value that specifies a range of planes or layers to be set the same; if either is omitted, the current plane or layer number is assumed. The numeric value after TO can be specified as "*" to indicate the highest numbered plane or layer. The "Clear" command clears the boundary condition from the facet(s).

To define boundary conditions in Electroconsolidation, the facets are divided into three groups: (1) facets in the base plane, (2) facets in the top plane, and (3) extrusion facets, which connect the planes. The operation commands are as follows:

"Boundary Conditions" → "Base plane" → "Select and define"

After Select and define, a dialogue box appears for the boundary condition definition.

Boundary Condition Modification		
Condition name		
Potential (*)		Radiation
Normal Magnetic		Tangential Magnetic
Normal Electric (*)		Tangential Electric
Symmetry		Clear
Value		
Other planes or layers		
From	To	All facets (*)
Accept		

With the power voltage V applied in Electroconsolidation, the Normal Electric boundary condition is assigned to all facets in the base and top planes. Then, after selecting the facets in the base plane of the lower ram, select potential, enter 0 (or -V/2) into the Value box, and select Accept. Repeatedly after selecting the facets in the top plane of the upper ram, select potential, enter V (or V/2) into the Value box, and select Accept. This nonzero scalar potential condition (a voltage V) is used to impose an external field (a potential/voltage difference across the Electroconsolidation system). In Electroconsolidation, the Tangential Electric boundary condition is used for the extrusion facets.

5.4 ELEKTRA Commands

This command creates an ELEKTRA format data file (see Table 1) that contains the preprocessor resident model:

Menu route: File Low Frequency (ELEKTRA)

Table 1. Commands that create an ELEKTRA format data file that contains the pre-processor resident model, and the default settings and functions of the commands

Command Parameter	Default	Function
FL	none	File name
UNITs	CGS	Units of the data CGS: practical CGS units METRe: SI units MICRon: SI units with lengths in _m MM: SI units with lengths in mm
ELEMent	LINE	Element-type over-ride LINEar: all elements linear QUADratic: all elements quadratic
ADAPtive	YES	Adaptive integration for RHSL YES or NO
MATE	LINEar	Material characteristics LINEar: fixed permeability NONLinear: permeability set by BH data
NIT	15	Maximum number of nonlinear iterations
TOLErance	0.001	Nonlinear iteration convergence tolerance
SOLUtion	SSAC	Solution type: SSAC: steady-state ac TRANsient: transient VELOcity: motion induced eddy currents
FREQuency	0	Frequency for SSAC Default value must be changed to zero (0) in the computation of Electroconsolidation.

ELEKTRA contains three analysis programs, one for steady-state ac (SSAC), one for calculating the transient response to fields switched on at zero time (TRANsient), and one to calculate eddy currents induced in moving components (VELOcity). The formats of the data for the three programs differ only slightly but the correct solution type must be chosen.

The ELEKTRA code obtains most of the information it needs from the parameters of the command. Before opening a data file, the program displays the current setting of the command parameters and gives the user the opportunity to change any of their values by typing new assignments, to accept the values by typing a null line, or to abort the creation file by typing "QUIT." The program then prompts the user to supply other needed information: properties (electrical conductivity and electrical permeability) for each material used, a title and additional frequencies (SSAC), or output times (TRANsient).

5.4.1 File Name, Units, and Element Type

ELEKTRA data files are ASCII files. The preprocessor adds a file name elek if none is given. ELEKTRA can interpret the geometric and material property data in any one of four unit systems:

Keyword	Units
CGS	Practical CGS units (cm,gauss,oersted,amp cm^{-2}, siemens cm^{-1})
METRE	Normal SI units (m, tesla, amp m^{-1}, amp m^{-2}, siemen m^{-1})
MM	SI units with mm (mm, tesla, amp m^{-1}, amp mm^{-2}, siemen mm^{-1})
MICRON	SI units with microns (μm, tesla, amp m^{-1}, amp μm^{-2}, siemen μm^{-1})

The element types (LINEar or QUADratic) are set during the Materials Definition Mode of the DEFINE command. The element-type override resets all elements to one type (LINEar or QUADratic). In Electroconsolidation, linear-element type is used for modeling analysis.

Keyword	Meaning
LINEar	All elements linear (8-noded)
QUADratic	All elements quadratic (10-noded)

5.4.2 Linear or Nonlinear Materials

ELEKTRA can solve problems with constant permeability (LINEar) or with a defined relationship between flux density and field intensity (NONLinear).

Keyword	Meaning
LINEar	Use constant permeability
NONLinear	Use permeability from B-H curve. The maximum number of iterations (NIT) and the convergence TOLErance can also be set.

In Electroconsolidation, all components are treated as linear materials. Therefore, there is no need for a B-H curve, NIT, and TOLErance.

5.4.3 Solution Type and Frequency

ELEKTRA can solve single-frequency, steady-state ac problems, where the response to the field from currents is switched on at time zero; or problems where eddy currents are induced in moving components. In Electroconsolidation, ELEKTRA solves all problems under steady-state ac with zero frequency.

5.4.4 Permeability and Conductivity

In the Materials Definition Mode of the DEFINE command, material names are supplied. For each name, ELEKTRA requires material properties, which depend on the type of analysis:

- For non-linear steady-state ac analysis, there are three prompts:
  1. The permeability is specified by a B-H characteristic. This is read from a file created by the B-HDAta command. The problem prompts the user to supply the name of a file for the material. If no file name is given, B-H is assumed.
  2. If the complex permeability hysteresis option is required, a value of the phase angle can be given. If no value is given, zero is assumed.
  3. A conductivity value must be supplied.
- For linear steady-state ac analysis, there are two prompts:
  1. A conductivity value must be supplied.
  2. A permeability value must be supplied. If the complex permeability hysteresis option is required, a value of the phase angle can be given as a second value in the input. If no value is given, zero is assumed.

In Electroconsolidation, linear steady-state ac analysis is used. Table 2 shows the permeability and conductivity values of all materials used in Electroconsolidation.

Table 2. Permeability and conductivity of all materials used in Electroconsolidation

Materials	Permeability	Conductivity (siemen/cm)
Preform (Advanced Ceramics YZ-110HS)	1.0	5.6E-14
Preform (Silicon Carbide)	1.0	0.19
Die or Ram (Graphite R7340)	1.0	837.6
Ram (Graphite R8650)	1.0	715.8
Reinforced Sleeve (Graphite Filcarb)	1.0	447.4
Insulation Felt (Flexible Graphite)	1.0	3.6
Insulation Coating	1.0	0.0
Fully Dense Insulation Silcon Carbide	1.0	0.24
Particulate Medium (Graphite)	1.0	f(ρ,P)

5.4.5 Title, Write, and End Commands

ELEKTRA data files can be annotated with a title, which can consist of as many lines of up to 80 characters as required. It is delimited by an * as the first character of the last line. Titles can be displayed by the post-processor SHOW command.

The END command stops the OPERA-3D preprocessor. All data files are closed.

Menu route: File → End OPERA-3d/Pre

If the program is ended without a "WRITE" command having been issued to store the pre-processor data, the file *oppre.backup* can be renamed with a file name extension *oppre*. This file is equivalent to a preprocessor data file. If it is not renamed, it may be replaced by subsequent use of the program.

Menu route: File → Write preprocessor file

The WRITE command writes an OPERA-3D preprocessor data file. There is one parameter, which defines the name of the file. If no file name extension is given, extension oppre is assumed.

Preprocessor data files consist of all the commands issued to the program, including cursor commands, except that some erroneous cursor hits are omitted. Comments are included among the cursor commands to help the user follow the sequence of commands. The character string CURS introduces cursor commands. Comments contain the character string **** followed by the number of the following comment lines.

The following is an example of the beginning of an OPERA-3d preprocessor data file.

```
================================================================

DEFI TYPE=MESH
XY
-13.081
0 10 0 10
****    1
Start of point definition
CURS P   3  .1525879E-04  .8000000E+03 -.3215156E+01  .1269267E+02
0,0,-13.081
3.81,90,-13.081
3.81,67.5,-13.081
3.81,45,-13.081
3.81,22.5,-13.081
3.81,0,-13.081
6.0325,90,-13.081
6.0325,67.5,-13.081
6.0325,45,-13.081
6.0325,22.5,-13.081
6.0325,0,-13.081
8.89,0,-13.081
8.89,22.5,-13.081
8.89,45,-13.081
8.89,67.5,-13.081
8.89,90,-13.081
20,90,-13.081
20,67.5,-13.081
20,45,-13.081
20,22.5,-13.081
20,0,-13.081
CURS
CURS Q   6  .1525879E-04  .8000000E+03 -.3215156E+01  .1269267E+02
****    1
```

```
Start of face definition
CURS C   9 .2050000E+03 .1980000E+03 .8394626E+00 .7597662E+00
CURS C   9 .1700000E+03 .1590000E+03 .1472107E+00 -.1329559E-01
CURS C   9 .2310000E+03 .1530000E+03 .1353707E+01 -.1322282E+00
CURS M  10 .2200000E+03 .1850000E+03 .1136142E+01 .5020790E+00
CURS F  11 .1525879E-04 .8000000E+03 -.3215156E+01 .1269267E+02
****    1
End of face
CURS C   9 .1580000E+03 .2210000E+03 -.9013285E-01 .1215674E+01
CURS C   9 .1590000E+03 .1640000E+03 -.7035422E-01 .8581490E-01
CURS C   9 .2070000E+03 .2070000E+03 .8790199E+00 .9381651E+00
CURS M  10 .1860000E+03 .2110000E+03 .4636687E+00 .1017453E+01
CURS F  11 .1525879E-04 .8000000E+03 -.3215156E+01 .1269267E+02
****    1
End of face
```

==========

5.5 Submitting a Job to ELEKTRA

The ELEKTRA program is composed of two parts: the first reads a data file and creates a direct-access database file; the second analyzes the data in the database file, storing the results in the database file and, optionally, in a text file. The ELEKTRA program will allow analysis of OPERA-3d data, creation of a database file, and reading of text result files. The database file is necessary for post-processing with the OPERA-3d post-processor.

To submit a job to ELEKTRA, enter the command "opera."

The ELEKTRA program will then prompt the user:

OPERA

Software Environment for Electromagnetic Design

2d or 3d processing or QUIT?
Enter: 3 (choose 3d processing)

OPERA-3d
Options are: PRE POST ELEKTRA TOSCA HELP or QUIT
Option:
Enter: ELEKTRA

Please select an option (SUBMIT, CONVERT, APPEND, IDEAS, HELP or QUIT):
Enter: S (submit the first job)

Submits an ELEKTRA data file to the solver for analysis.
Please give ELEKTRA data filename (without the .ELEK suffix)
Enter: test1 (name of the first job)

Steady-state AC (ss), Transient (tr) or Velocity (vl)?
Enter: ss (always steady-state AC with frequency 0 in Electroconsolidation)

Do you want to run the analysis now or later? (n or l)
Enter: l (run the analysis later)

File ELEKTRA.batch contains commands to run analyses
Please select an option (SUBMIT, CONVERT, APPEND, IDEAS, HELP or QUIT):
Enter: s (submit the second job)

Submits an ELEKTRA data file to the solver for analysis.
Please give ELEKTRA data filename (without the .ELEK suffix)
Enter: test2 (name of the second job)

Steady state AC (ss), Transient (tr) or Velocity (vl)?
Enter: ss

Do you want to run the analysis now or later? (n or l)
Enter: l (submit the job later)

Do you want a new deferred execution file? (y or n)
Enter: n
(Jobs 1 and 2 are in one deferred execution file. Job 2 will be executed after completion of Job 1)

File ELEKTRA.batch contains commands to run analyses
Please select an option (SUBMIT, CONVERT, APPEND, IDEAS, HELP or QUIT):
Enter: q (quit the process of the submitting)

Options are: PRE POST ELEKTRA TOSCA HELP or QUIT
Option:
q (out of opera process)

Once the ELEKTRA.batch file is created, the users can simply type

ELEKTRA.batch

to start running the job.

6 Post Processor OPERA-3D of ELEKTRA

6.1 Introduction

The OPERA-3d postprocessor displays and performs further calculations on results from electromagnetic field analysis programs, including ELEKTRA. The analysis programs use finite elements to model 3-D electromagnetic devices. The postprocessor provides facilities to view the finite-element data with superimposed contours of results, and to process and display the results calculated along lines or on 2-D areas, in addition to calculating specific functions such as resistivity heat and trajectory.

The OPERA-3d postprocessor can be used interactively or in a batch stream. The program contains features that are best suited to color workstations or X-terminals, although it can be used on terminals with less functionality. In batch mode the graphics instructions can be stored in a file to be viewed later.

The program is used to display 3-D finite-element models from direct-access database files created by the analysis programs. Many results in the database can be available to the program through the ACTIVATED command at any one time. Reflection codes and symmetries can also be given to replicate the finite-element mesh so the complete model is available to OPERA-3d, even if only a small section of it was analyzed and the rest was implied by boundary conditions.

After device nomination, or when the OPERA-3d postprocessor is restarted with the CLEAR command, the program looks for a file called opera.comi in the current file directory. If such a file exists it is read into the program as a $ COMInput file. This allows the user to reset the default values of certain commands, e.g. COLOUR and UNITS or define frequently used CONStants and PARAmeters each time the program is started.

Two methods of command and data entry are available: (1) menu system or GUI (graphical user interface), in which command selection and data specification are carried out under mouse control, and (2) command line input, in which command selection and data specification are carried out from the keyboard. Under normal operation, the preprocessor starts in the GUI mode. Only the menu system or GUI is introduced in this guide.

6.2 Reference Guide to Menu Mode

FILE	UNITS	OPTIONS	DISPLAY	HELP	CONDUCTORS	FIELDS	MENU_OFF

FILE	
Database files	
Activate database files	Make a results database available to program
Re-load active file	Read in a previously loaded file
List active files	List previously activated files
Commands in	Read command input file
Output message mode	Set the message display option
Change directory	Change the current directory
System command	Operating system commands
Return	Close Menu
End OPERA-3d/Post	End postprocessing session

UNITS	
SI Units	Set working units to SI
CGS Units	Set working units to CGS
Imperial Units	Set working units to Imperial
Length unit	Specify working units to SI
Flux density units	Specify flux density units
Magnetic field unit	Specify magnetic field strength units
Electric field unit	Specify electric field strength units
Scalar potential unit	Specify scalar potential units
Vector potential unit	Specify vector units
Conductivity unit	Specify conductivity units
Power unit	Specify power units
Force unit	Specify force units
Energy unit	Specify energy units
Return	Set current units and close menu

OPTIONS	
Options	
Calculator	
Parameters	Set a user-defined parameter
Constants	Set a user-defined constant
Color settings	Modify colors in graphics palette
Clear and Reset	Clear all mesh data and restart
Dump picture	Send current graphics screen to a file
Graphics output	Select type of graphics output
Title options	Specify title, data, and extra data
Title position	Specify title position in window
Display title	Enter text of title
Return	Close menu

DISPLAY	
Select and Display	
Select parts to display	Specify parts of model to display
Display selected parts	Display selected parts of model
Refresh display	Clear and redraw graphics window
Return	Close menu

HELP	
System overview	General description of postprocessor
Menus	Assistance with the menu system
Parameter boxes	Use and editing of parameter boxes
Function keys	Description of function key operation
Message boxes	Information and use of message boxes
Command line	Assistance with command line entry
System variables	Lists system variables
Return	Close menu

CONDUCTORS	
Conductors	(Not used in the Electroconsolidation)
Define a conductor	Start creating a new conductor
Erase conductors	Delete conductor
Modify conductors	Modify parameters of conductors
Print data	Display conductor's data
Write data to a file	Store conductor data
Return	Close menu

FIELDS	
Field calculations	
Calculation options	Select calculation method
Field components	Enter components to be displayed
Fields at a point	Evaluate field at third coordinate
Fields on a line	Evaluate field along a line
Fields over a patch	Evaluate fields over a 2-D surface
Integrals	Integrate fields over objects etc.
Fields harmonics	Fit harmonic coefficients to a line etc.
Return	Close menu

6.3 DISPLAY Command

The display command DISP draws pictures of the 3-D geometry of the finite-element mesh and conductors. Pictures can be line drawings or can show colored surfaces, with the hidden surfaces obscured. It is also possible to overlay the geometry with displays of the field quantities.

Scalar field quantities or single components of vector field quantities can be displayed as line contours or colored zones. Vector field quantities can be displayed as vectors at the center of the element faces. Vectors also indicate the current directions in the conductors.

The parameter controls the coordinate limits of the volume of the 3-D space included, the direction of the view, and how the selected surfaces should be displayed.

The system variables used in Electroconsolidation are listed below in groups. Unit conversion is performed on the basic system variables. The expressions listed below will give correct values for derived system variables only if consistent units are used.

• Field point coordinates

X, Y, Z	Field point coordinates
TX, TY, TZ	Tangential unit vector to lines
NX, NY, NZ	Normal unit vector to surfaces

• Potential and field values at field point

POT	Magnetic or electric scalar potential
BX,BY,BZ	Magnetic flux density
BMOD	Magnitude of magnetic flux density: SQRT(BX**2+BY**2+BZ**2)
HX,HY,HZ	Magnetic field strength
HMOD	Magnitude of magnetic field strength: SQRT(HX**2+HY**2+HZ**2)
DX,DY,DZ	Electric flux density
DMOD	Magnitude of electric flux density: SQRT(DX**2+DY**2+DZ**2)
EX,EY,EZ	Electric field strength
EMOD	Magnitude of electric field strength: SQRT(EX**2+EY**2+EZ**2)
JX,JY,JZ	Current density
JMOD	Magnitude of current density: SQRT(JX**2+JY**2+JZ**2)
AX,AY,AZ	Magnetic vector potential
AMOD	Magnitude of magnetic vector potential: SQRT(AX**2+AY**2+AZ**2)

• Material properties at field point

MU	Isotropic permeability, BMOD/HMOD
EPSILON	Permittivity, DMOD/EMOD
SIGMA	Conductivity

• Solution value

FREQ	Frequency (steady-state ac)

• Constants

PI	π
MU0	permeability of free space (SI)
EPSILON0	permittivity of free space (SI)

Not all field values apply to all problems, and the precise meaning of the scalar potential depends on the potential type of the element concerned (TOTAL magnetic scalar potential, REDUCED magnetic scalar potential, or electric scalar potential in VECTOR regions or electrostatics problems). In the applications of Electroconsolidation,

$$\text{Electrical resistivity heat} = \sigma E \bullet E = \sigma (EX*EX + EY*EY + EZ*EZ);$$

Therefore, only electrical field strength components EX, EY, and EZ are the required output data to be transferred from ELEKTRA to MaPS.

6.4 UNIT Command

The UNITS command sets the units to be used to interpret user input and to display geometric and field data. Each of the parameters can be set to one of a set of predefined character strings that corresponds to commonly used units.

Menu Route: UNIT

The command line parameters, their possible units, default units, and functions are listed in Table 3.

Table 3. Command line parameters of UNIT command, their default and possible units, and functions.

Command Parameter	Default	Possible Units	Function
LENGTH	CM		Unit of length
		CM	Centimeter
		IN	inch
		METRE	metre
		MICRON	micron
		MM	millimetre
FLUX	GAUS		Unit for magnetic or electric flux density
		CM2	coulomb metre^{-2}
		GAUSS	gauss
		KGAUSS	kilogauss
		TESLA	tesla
FIELD	OERS		Unit for magnetic field strength
		AM	ampere metre^{-1}
		OERSTED	oersted
SCALAR	OCM		Unit for scalar potential
		AMP	Ampere
		OCM	oersted centimeter
		VOLT	volt
VECTOR	GCM		Unit for vector potential
		GCM	gauss centimeter
		WBM	weber meter^{-1}
CONDUCTIVITY	SCM		Unit for conductivity
		SCM	siemen centimeter^{-1}
		SIN	siemen inch^{-1}
		SM	siemen meter^{-1}
		SMM	siemen millimeter^{-1}
		SMU	siemen micron^{-1}

Table 3. Command line parameters of UNIT command, their default and possible units, and functions. (CONTINUED)

Command Parameter	Default	Possible Units	Function
CURD	ACM2		Unit for current density in conductors
		ACM2	ampere centimeter^{-2}
		AIN2	ampere inch^{-2}
		AM2	ampere metre^{-2}
		AMM2	ampere millimeter^{-2}
		AMU2	ampere micron^{-2}
POWER	WATT		Unit of power
		ERGS	erg second
		HP	horse power
		WATT	watt
FORCE	NEWTON		Unit for force
		DYNE	Dyne
		GRAMME	gramme force
		KG	kilogram force
		LBF	pound force
		NEWTON	newton
ENERGY	JOULE		Unit of energy
		BTU	British thermal unit
		ERG	erg
		JOULE	joule
ELECTRIC	VCM		Unit for electric field strength
		VCM	volt cm^{-1}
		VIN	volt in^{-1}
		VM	volt meter^{-1}
		VMM	volt millimeter^{-1}
		VMU	volt micron^{-1}

6.5 Options Commands

6.5.1 TITLE Command

Menu Route: OPTIONS → Title

The command line parameters, their possible units, default units, and functions are listed in Table 4.

Table 4. Command line options of TITLE command, their default and possible options, and functions.

Command Option	Default	Abbre-viation	Function
STRIng	none	none	
POSItion	TL		Graphics window title position
		BC	Bottom center
		BL	Bottom left
		BR	Bottom right
		TC	Top center
		TL	Top left
		TR	Top right
KEEP	YES		Title preservation switch
		NO	Only display title once
		YES	Display title on subsequent pictures as well
NOW	YES		First appearance switch
		NO	Display after next graphics window clear
		YES	Display immediately
DATE	YES		Date, time, and page number switch
		NO	Date, time, and page number not displayed
		YES	Date, time, and page number displayed
EXTRa	YES		Extra information switch
		NO	Extra information not displayed
		YES	Extra information displayed

The TITLE command controls the display of titles and other information on the graphics window. The items that can be controlled are an additional title, and the default labelling of the graphics window with date, time, and page number.

6.5.2 DUMP Command

Menu Route: OPTIONS → Dump picture

The command line parameter of DUMP command and its function are listed in Table 5.

Table 5. Command line parameter of DUMP command and its function.

Command	DUMP	
Parameter	Default	Function
File Name	none	Name of file to contain the picture commands

The DUMP command dumps the graphics commands used to create the current display to a file. If no file name extension is given, and extension *ps* is assumed.

6.6 HELP Command

Menu Route: HELP

The program prompts for the name of a topic:

HELP
System Overview Menus Parameters Boxes Function Keys Message Boxes File Boxes Command Line System Variables QUIT

The HELP command provides help to remind users of several program features. The topics are:

• System Overview that gives a flow chart of the OPERA-3D postprocessor top-level commands that indicate the sequence in which they should be used to obtain analysis results.

• Menus

• Parameter Boxes

• Function Keys

• Message Boxes

• File Boxes

• Command Line, which summarizes the syntax and built-in help features of the command decoder, including details of subcommands and cursor commands.

• System Variables lists the system variables and their meanings.

6.7 END Command

Menu Route: File → End OPERA-3D/Post

The END command contains no command line parameters; it stops the OPERA-3D post-processor. All data files are closed.

Command	END
No Parameters	

7 Data Transfer from ELEKTRA to MaPS

The OPERA-3D post-processor reads/displays data from the direct-access database files created by the analysis programs. Other files can be created and read by the postprocessor for use by the result display commands.

Database results files contain all the information that is used and calculated by the analysis programs, and includes the finite-element mesh, the boundary conditions, the material properties, and the solution options for Electroconsolidation applications. These files can only be read by the analysis programs and the postprocessor.

7.1 Process of Data Transfer

The TABLE command is provided to facilitate an interface to other programs (e.g., MaPS Code in Electroconsolidation). It performs three tasks:

1. Reads an input data file that should contain field point coordinates and can also contain other columns of values (see Table 6).

2. Calculates field values at the points and outputs for up to 12 expressions of the coordinates or field values for each point.

3. Writes an output data file that contains the values calculated (e.g., values of resistivity heat in Electroconsolidation).

Table 6. Command line parameters of TABLE command, their default values and functions

Command	TABLE	
Parameter	Default	Function
INFIle	none	Name of input data file
OUTFile	none	Name of output data file
F1	X	Expression for first column in output data file
F2	Y	Expression for second column in output data file
F3	Z	Expression for third column in output data file
F4	none	Expression for fourth column in output data file
F5	none	Expression for fifth column in output data file
F6	none	Expression for sixth column in output data file
F7	none	Expression for seventh column in output data file
F8	none	Expression for eighth column in output data file
F9	none	Expression for ninth column in output data file
F10	none	Expression for tenth column in output data file
F11	none	Expression for eleventh column in output data file
F12	none	Expression for twelfth column in output data file

The user must assign a name to each input file INFI and output file OUTF. The expressions (F4 - F12) can use as variables any of the system variables and any user variables (e.g., define F4 = resistivity heat = σ E • E, in Electroconsolidation). They are evaluated at each field point (defined by F1, F2, and F3) in the input datafile. All of the values for one point appear in one record of the file.

7.2 Programming a Data Transfer File (*.comi)

To transfer the data of electrical resistivity heat from ELEKTRA to MaPS, the user must go to the OPERA/3D postprocessor of ELEKTRA, open the output file *.ELEKB (or *.elekb) of ELEKTRA, and then command in the *.comi file to read the values calculated by ELEKTRA. The *.comi file created by the TABLE command is shown as follows:

```
/ UNIT (centimeter)
/
$PROMPT INP 'File to contain coordinates for cube'
$OPEN 1 &INP&.table WRITE -REDI
$FORM 1 EXPO 13
$FORM 2 STRI 33 STRI='    X         Y         Z'
$ASS 2
$WRITE 1
$ASS 1 1 1
/
  $WRITE 1   0.2814E+00   0.2814E+00   -0.2965E+01
  $WRITE 1   0.2814E+00   0.2814E+00   -0.2540E+01
  $WRITE 1   0.2814E+00   0.2814E+00   -0.2115E+01
  $WRITE 1   0.2814E+00   0.8441E+00   -0.2965E+01
  $WRITE 1   0.2814E+00   0.8441E+00   -0.2540E+01
  $WRITE 1   0.2814E+00   0.8441E+00   -0.2115E+01
  $WRITE 1   0.8441E+00   0.2814E+00   -0.2965E+01
  $WRITE 1   0.8441E+00   0.2814E+00   -0.2540E+01
  $WRITE 1   0.8441E+00   0.2814E+00   -0.2115E+01

  $WRITE 1   0.4232E+01   0.1048E+02   0.2965E+01
  $WRITE 1   0.4232E+01   0.1048E+02   0.3575E+01
  $WRITE 1   0.4232E+01   0.1048E+02   0.4375E+01
  $WRITE 1   0.4232E+01   0.1048E+02   0.5245E+01
  $WRITE 1   0.4232E+01   0.1048E+02   0.6562E+01
  $WRITE 1   0.4232E+01   0.1048E+02   0.8255E+01
  $WRITE 1   0.4232E+01   0.1048E+02   0.9948E+01
/
$CLOSE 1
/
$PROMPT OUTP 'File to contain EMF data and coordinates'
$PARA #QEMF SIGMA*(EX*EX+EY*EY+EZ*EZ)
TABLE &INP& &OUTP&  F4=#QEMF
/
```

8 Geometry Preprocessor CPRE

8.1 Instructions and Guidelines

The geometric preprocessor generates a neutral file from text input. The neutral file appears in the current directory as PROJECT.NE, where PROJECT is the currently selected project name. CPRE also generates AVS ucd files in a directory that contains the PROJECT name. These files can be used to visualize the input, which, for the geometric preprocessor, is in a free-form text format (see samples in Appendix B). Keywords and fields are separated by delimiters.

Valid keyword entities are

"g" grid
"l" line
"tr" triangle
"p" patch
"h" hyperpatch
"n" named component

Valid delimiters between keywords and fields can be

" " space
"," comma
"=" equals
"(" and ")" parentheses.

GRIDS

A grid is a point in space and consists of an identifier, id# (integer), and x,y,z coordinates (3 reals). Grids must be defined on one line. Any keyword that starts with a "g" will be processed as a grid.

The format is

g id# x_coordinate y_coordinate z_coordinate,

for example,

grid(23) = (100.0, -50., 20.)
g 23 100 -50 20.
grid 23 = (100,-50,20),

which will all be interpreted as grid id = 23 x = 100. y = -50. z = 20.

LINES

A line is a straight line in space. Here, it consists of an identifier id# (integer), and two grid ids (two integers). Lines must be defined on one line. Any keyword that starts with "l" will be processed as a line.

The format is

```
l id# grid_id1 grid_id2,
```

for example,

```
line(51) = (18, 2)
l 51 18 2
line 51=18,2,
```

which will all be interpreted as line id = 51, with the end points grid(18) and grid(2).

TRIANGLES

A triangle is a plane surface in space. Here, it is a tri element and consists of an identifier, id# (integer), and three grid ids (three integers). The three grids should not be colinear. The burden is on the user to input a noncolinear sequence. Triangles must be defined on one line. Any keyword starting with "tr" will be processed as a triangle.

The format is

```
tr id# grid_id1 grid_id2 grid_id3,
```

for example,

```
triangle(51) = (18, 2, 44)
tr 51 18 2 44
tri 51=18,2,44,
```

which will all be interpreted as a triangle id = 51, with vertices grid(18), grid(2), and grid(44).

PATCHES

A patch is a surface in space. Here it is a quad element and consists of an identifier, id# (integer), and four grid ids (four integers). The four grids must be defined such that straight lines that connect grid_1 to grid_2 to grid_3 to grid_4 back to grid_1 don't cross. The burden is on the user to input the correct sequence.

```
Valid: 1 - 2  or  1 - 4        Invalid: 1 - 2  or  1   3
       |   |      |   |                      X          |X|
       4 - 3      2 - 3                  3 - 4      4   2.
```

Patches must be defined on one line. Any keyword starting with a "p" will be processed as a patch.

The format is

p id# grid_id1 grid_id2 grid_id3 grid_id4,

for example,

patch(51) = (18, 2, 44, 9)
p 51 18 2 44 9
pat 51=18,2,44,9,

which will all be interpreted as patch id = 51 with vertices grid(18), grid(2), grid(44), and grid(9).

HYPERPATCHES

A hyperpatch is a volume in space. Here it is a hex element and consists of an identifier, id# (integer), and eight grid ids (eight integers). The eight grids are defined such that the first four points out of the volume in a right-handed, counterclockwise fashion, and grids 5 - 8 are above 1 - 4, respectively. The burden is on the user to input the correct sequence.

Valid: below above
1 - 2 5 - 6
| | | |
4 - 3 8 - 7.

Hyperpatches must be defined on one line. Any keyword starting with an "h" will be processed as a hyperpatch.

The format is

hp id# grid_id1 grid_id2 grid_id3 grid_id4 grid_id5 grid_id6 grid_id7 grid_id8,

for example,

hp(17) = (18, 2, 44, 9, 8, 12, 14,20)
hp 17 18 2 44 9 8 12 14 20
hyperpatch 17=18,2,44,9,8,12,14,20,

which will all be interpreted as hyperpatch id = 17 with vertices grid(18), grid(2), grid(44), grid(9), grid(8), grid(12), grid(14), and grid(20).

NAMED COMPONENTS

A named component is a grouping of geometric entities with a user-defined name. Here it consists of a NAME (string), type (keyword), and a list of ids (integers). Although the list contains number of items, all of the items must be of the same type (i.e., grid, line, triangle, patch, or hyperpatch). The user-defined NAME is limited to 12 characters and all should be capitals to be consistent with PATRAN. The id_list can span several lines.

The format is

n NAME type id_list,

Examples:

n MESH_CONTROL g 3 6 8 34
name = MESH_CONTROL grid(3, 6, 8, 34),

both of which define MESH_CONTROL as four grids.

Any line not recognized by a keyword is treated as a comment. Comments can also be placed on the same line after the complete specification of a keyword entity that requires a one-line spec.

8.2 CPRE Shell Scripts

```
-----------------------------------------------
| Geometry Preprocessor
-----------------------------------------------
Enter new project name : test
Add project -->test<-- (OK) :
---------------------------------------
CPRE -->test<-- What now?
---------------------------------------
1) Edit input dataset
2) Process input dataset
3) Avs to view geometry
4) Neutral view via PATRAN
5) Change project
6) Delete project
7) Output
8) Help
9) Quit (*)
Choose (9) : 1
Enter input pre dataset (?) : ?
Choose file for INPUT or Enter "n" for new :
There is only one to choose -->MPs-11.pre.1<-- (OK) :
---------------------------------------
CPRE  -->test<--
INPUT -->MPs-11.pre.1<-- What now?
---------------------------------------
```

```
1) Edit input dataset
2) Process input dataset
3) Avs to view geometry
4) Neutral view via PATRAN
5) Change project
6) Delete project
7) Output
8) Help
9) Quit (*)
Choose (9) : 2
Process input -->MPs-11.pre.1<--
 ______________________________
|  test.NE created!.! !   |
 ------------------------------
 CPRE  -->test<--
 INPUT -->MPs-11.pre.1<-- What now?
 ---------------------------------------
1) Edit input dataset
2) Process input dataset
3) Avs to view geometry
4) Neutral view via PATRAN
5) Change project
6) Delete project
7) Output
8) Help
9) Quit (*)
Choose (9) : 3
AVS SELECTED
 ---------------------------------------
 CPRE  -->test<--
 INPUT -->MPs-11.pre.1<-- What now?
 ---------------------------------------
1) Edit input dataset
2) Process input dataset
3) Avs to view geometry
4) Neutral view via PATRAN
5) Change project
6) Delete project
7) Output
8) Help
9) Quit (*)
```

9 Geometry Initialization of MaPS

9.1 Introduction

The first step necessary to run the simulation is to set up the geometry. Then, going down the flow chart, either the geometry can be meshed or boundary and initial conditions (BCIC) can be set for the geometry. As indicated earlier, CPRE can be used to create the geometry and thus a neutral file that consists of named components that are read by MaPS. In the following section, we refer to the problem of setting the initial and boundary conditions for each named component. Two methods are available for initializing various components of the geometry. The Unix editor can be used to manually create a *BC file that is then assigned to a BCIC.*. The BC file must contain all of the named component initializations such as temperature. The BC file is then read and combined with the MESH.* files by MaPS via the Bcgen module to initialize the simulation and store the data in RUN.0. Another way to initialize is to use the visual interface of MaPS for boundary condition setup. This Vbounds interface helps to visualize various parts of the geometry with differing colors, thereby building confidence that proper physical attributes will be assigned during the creation of the geometry. The boundary and initial conditions can be set in the visual interface, which then creates the BCIC.* input file that assigns physical attributes to the geometry.

9.2 Visual BCIC

The visual interface for boundary condition setup is a user-friendly tool capable of ascertaining whether proper geometry is created with CPRE. Incremental zoom-in, zoom-out, spinning, rotating, etc., is difficult while using CPRE. The Vbounds module of the MaPS software package allows the user to do this and much more while setting up the boundary and initial conditions.

Because the Vbounds module uses the CPRE libraries, it is important that these libraries be installed into the system. The display environment should also be set to accommodate the HOOPS windows. As described in Section 11 (Input and Output Data of MaPS) of this document, the user has a choice of selecting the visual BCIC or merely assigning the input. The user must either have created the *BC file already or must have run Vbounds at least once, setting up the appropriate boundary conditions and thus creating the *BC file they can then use to assign input.

In this section, we try to explain the various aspects of Vbounds so initial and boundary conditions can be set up for the geometry in question. As explained in the GEOMETRY assignment section of this user guide, the geometry consists of volumetric hyperpatches and surface patches. Thus, the initial conditions are to be set on volumetric hyperpatches alone and boundary conditions should be set on the boundary surfaces, which are made up of patches. Initial and boundary conditions cannot be set on grids. For example, MESH_CONTROL is a named component used to force specific grid planes in meshing the geometry. Clearly, assigning initial or

geometric conditions on grids is irrelevant and therefore no conditions can be assigned to MESH_CONTROL in the Vbounds module.

When Vbounds is first invoked, following buttons appear on top of the screen. Each button has a different meaning and implication. In the sections that follow, we discuss each of the buttons in more detail.

File	Boundary Conditions	Preferences	View	Help	Note

9.2.1 File

The File button offers the user the following options:

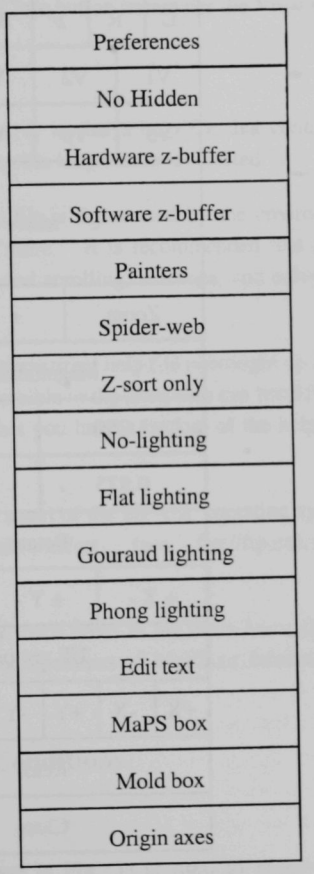
File
Read Geometry
Read Components
Save
Print
Quit

Thus, selecting the File button allows the user to read geometry or named components from the neutral file. Once the boundary conditions are set, we can press the Save button to save a file that contains the current boundary condition parameters. The Print button can be pressed if the currently active geometry display on the screen is to be printed. The Vbounds module can be terminated by pressing the Quit button, in which case the user must confirm the quit:

Cancel Quit
Confirm Quit

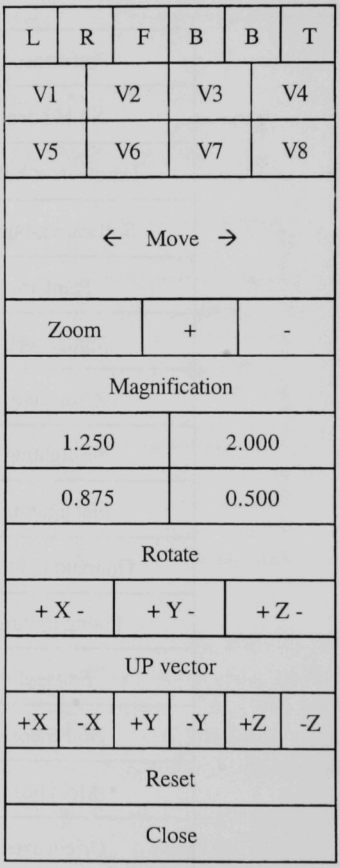
9.2.2 Preferences

It is possible that when the named components are colored, the geometry hyperpatches may appear cut and not coherent with each other. To make the view more appealing to the eye, depress the Preferences button in the main menu. Once the Preference submenu appears, several choices are available for the display of the named components. The submenu allows the user to specify options for hidden-surfaces rendering, lighting, editing, and some visibility:



9.2.3 View

When the View button is invoked, the view submenu (shown below) appears. This submenu lists the various viewing aspects of the geometry and appears on the right side of the screen. Specific views can be selected by selecting Left (L), Right (R), Front (F), Back (B), Bottom (B), Top (T), View1 (V1), View2 (V2), ...View7 (V7), or View8 (V8).



Pressing the mouse pointer on the + of the Zoom button zooms into the geometry, and pressing the mouse pointer on the - of the Zoom button zooms out of the geometry. Pressing the Magnification button magnifies or reduces the geometry or the named components, based on the selected ratio of 1.250, 2.000, 0.875, or 0.500.

Activating the Rotate button spins the geometry in the plane of the screen. If at any time, the geometry view is to be restored, the Reset button is pressed. This displays the original view, as seen when Vbounds was first initiated. It may be noted that after zooming in and out, the various view buttons can still be pressed to display various angles of the zoomed-in or the zoomed-out section of the geometry. Pressing the Close button terminates the View submenu.

9.2.4 Help and Notes

The help facility is context sensitive in that a help file that contains information about the current window is brought up whenever the help button is selected.

The pager used to view the help file is determined by the environmental variable PAGER. On HP-UX systems this defaults to "more." It is recommended that this be changed to "less," which allows both forward and backward scrolling, searches, and editing. The help window can be closed by entering "q."

By entering "v" while in "less," the current help file is brought up in the vi editor. If the help files in CAPSBASE/doc are write accessible to the user, who can modify or add to these help files. Before this feature is used, be sure that you have a backup of the help files because the contents can easily be deleted.

Although "less" is generally not a part of the HP-UX operating system, it is available on the WWW and compiles with little effort. (see ftp://ftp.cae.wisc.edu/hpux9/Users/less-237.magic.1.tar.gz).

Selecting "Notes" opens the file maps.notes in the users home directory with the vi editor. The user can add notes reverent to the operation of MaPS or information about the simulations being performed.

9.3 Initial and Boundary Conditions

9.3.1 Initial Conditions Form

The "Initial Conditions Form" (shown in Fig. 3) is used to initialize named components that consist of hyperpatch volumes only. Thus, CAPSGEOMETRY can be initialized if the entire domain is to have identical initializations. Note that in using MaPS software to model the

Electroconsolidation process, all of the components are always solid; thus, the fraction filled for each component is 1.0.

The material must be specified for each named component. Pressing the Material... button on the Initial Conditions Form activates the list of materials in the properties database of MaPS. For further information on adding or editing the materials database files, refer to Section 12 (Material Properties of MaPS). The material of interest can then be chosen from the list, and the named component is assigned that material.

The Initial Conditions Form is obtained when CAPSGEOMETRY is pressed in the menu under "Boundary Conditions." If the activated named component has thermal structures assigned to it, those buttons are to be activated, thus invoking menus to initialize them. In Fig. 3, CAPSGEOMETRY, preform, powder, die, ram, filcarb, and felt can be initialized in this manner because they are made up of hyperpatch volumes alone. Refer to the section on thermal structures (Section 9.6) for a theoretical explanation.

Once the preferred named component is pressed in the menu under Boundary Conditions, an asterisk appears in the box adjacent to the colored box that corresponds to that named component. Simultaneously, the named component is painted with that color and the related initialization form appears on the left side of the screen. If only the named component is to be viewed, the initialization form can be deactivated as follows. First, maneuver the mouse pointer to the box adjacent to the colored box. Then, press the mouse button to activate editing of the box. Entering the asterisk in the box, pressing return, and then pressing the Apply button at the lower right of the menu colors the named component without activating the initialization box. This is useful if only the named components are to be viewed and the initialization is either not to be conducted or has already been done. In the following discussion of thermal structure setup, a step-by-step explanation is given of the various windows that are encountered. We describe the various screens that appear when the Vbounds module is activated to assign initial and boundary conditions to the geometry. A three-dimensional Electroconsolidation problem is illustrated as an example.

When Vbounds is first initiated in the MaPS scripts, Fig. 4 appears and essentially displays the problem geometry. The geometry is displayed according to the neutral file created by CPRE. The various aspects of the buttons shown in Fig. 4 are defined in Section 9.2, the Visual BCIC section of this user guide. The Boundary Conditions button brings up the screen shown in Fig. 5; this is essentially a subwindow that lists all named components as described in the geometry and according to the neutral file that is read. The indices are initially zero. To allow setup of initial and boundary conditions for a geometric section, the corresponding number must be other than zero.

Setting the indices allows us to assume that the initial conditions for the preform are to be set when the PREFORM button on the screen shown in Fig. 5 is pressed; the button color then changes and Fig. 6 appears. Now, the material, phase, pressure, fraction filled, etc., can be specified for the specified component (in our case, the PREFORM).

The Material... button gives the display shown in Fig. 7. This display lists of all the material property database files that exist in the materials_library directory. The names of these files are included in the available_materials file in the materials_library directory. The material can be selected by depressing the corresponding button (which then changes color), and then pressing return. Then, the display shown in Fig. 6 appears with the Material... box filled with the name of the selected materials database file in place of the "unspecified" entry.

At this point, the specification for the geometric section PREFORM is complete. Similarly, other geometric sections can be initialized. The time required to set the initial condition is minimal (on the order of seconds), and the procedure is extremely systematic. Thus, the user is encouraged to use the Vbounds module rather than create the *BC file by using the vi editor, which is likely to lead to mistakes.

9.3.2 Boundary Conditions Form

The Boundary Conditions Form (Fig. 8) is similar to the Initial Conditions Form but is restricted to setting up the boundary conditions on the surface only. This form has various subwindows, each of which is targeted toward the particular surface that must be assigned certain boundary conditions.

If the named component is related to a boundary wall, the Wall submenu must be depressed to activate and tell MaPS that the surface is a wall. Various boundary conditions, such as Freeslip, Noslip, Thermal Structure, Constant Temperature, and Adiabatic can be ascribed to the surface walls. A constant wall may suggest prescribing an (initial) temperature. If no walls are defined, the surface boundaries are considered to be adiabatic.

A Symmetry choice suggests that one-half of the geometry is such that the other half is its mirror image. The existence of the symmetric surface reduces the geometry by one-half and thereby significantly reduces the simulation time. The Symmetry choice assumes that no wall is present and that the particulate medium is as if both parts exist in the geometry. Thus, the temperature is initialized, depending on the temperature of the adjacent cell.

Thermal structures can be defined by depressing the Thermal Structure button. A Thermal Structures List window, shown in Fig. 9 appears. Here, because no settings are in place yet, pressing the NEW button provides the window Define Thermal Structure in Fig. 10. After determining the Area Multiplier (default value: 1) and Temperature (change default value of 20.0 to room temperature 25), pressing the Material button in Fig. 10 activates the list of materials in the properties database of MaPS. Then, material property "constant_t" is assigned to ambient air; and both Number of partitions and Partition size of the ambient air are entered for the Electroconsolidation application.

Now, everything is initialized except the Heat Transfer Coefficient, which provides information required in the calculation of the surface heat transfer coefficient in thermal structure models. When the Heat Transfer Coefficient button shown in Fig. 10 is selected, the window shown in Fig. 11 appears. Here, because no settings are in place yet, the Heat Transfer Coefficient List is empty. Pressing the New button gives us the window in Fig. 12, from which the Radiation Heat Transfer Flag (default 0.0), the Heat Transfer Coefficient Multiplier (default 1.0), and the Characteristic Length (default 6.35E-03) are to be initialized. Although, default values are given, they can be changed at the user's discretion. For example, the Radiation Heat Transfer Flag should be set to 1.0 in Electroconsolidation. The Heat Transfer Correlation button must now be pressed to assign the correlation to the coefficients and effect the appearance of the window shown in Fig. 13. Here again, because nothing is yet defined, the Heat Transfer Correlation List is empty. Pressing the NEW button produces the window shown in Fig. 14. Now, the user can name the correlation, briefly describe it, and assign parametric values to correlation coefficients C1, C2, C3, C4. The defaults are automatically given and the default name, "generic_htc," is also assigned. The user can either use the default values or alter the values. In Electroconsolidation, C1 should be set to 25.0 for natural convection and C2, C3, C4 are not used because the ambient air is assumed to be free-flowing in a huge reservoir. Saving the correlation datalist by selecting Save allows the user to move one step up in the hierarchy and brings up the window shown in Fig. 15. Pressing Return gives the window shown in Fig. 16.

Now that the Heat Transfer Correlation is defined, the definition of the heat transfer coefficient is complete. Saving the definition by selecting Save, we obtain the window shown in Fig. 17. Now, thermal structure assignment is complete, as is the entire parametric listing shown in Fig. 18. Saving the definition by selecting Save gives us the window shown in Fig. 19.

Again, because one data list has been created, the Thermal Structures List contains one data list file named "generic_ts." Note that we have altered the default values and have accepted all of the parametric values, file names, and descriptions provided by the MaPS software. Now, pressing "return" displays the window shown in Fig. 20.

At this point, the specification for the geometric section RAMS (the side surfaces of the rams) is complete. Similarly, other geometric sections can be initialized. The time required to set the boundary condition, including thermal structures, is minimal (on the order of seconds), and the procedure is extremely systematic. Thus, the user is again encouraged to use the Vbounds module rather than create the *BC file by using the vi editor, which is likely to lead to mistakes.

In Electroconsolidation, to account for the cooling reservoirs that exist at the ends of rams, Noslip and Constant Temperature (50°C) are assigned to the ends of both upper and lower rams for boundary conditions (Fig. 21). Thermal Structure boundary conditions are assigned to die, ram, or felt surfaces that are next to ambient air (Figs. 8-20). Freeslip and Adiabatic boundary conditions are considered for central surfaces for symmetry (Fig. 22).

9.4 Alternative BCIC Assignment

The Vbounds module helps identify various parts of the geometry and set up the various initial and boundary conditions. This module, when exited, saves the input for BCIC.* and optionally to a BC file. It is possible to create the same BCIC.* input file if we know where the various geometric sections are physically located so various initial and boundary conditions can be written down for each named component. Following are two of the various terms that have meanings in the visual interface of MaPS to set up the initial and boundary conditions:

vof	Volume of particulate medium filled
temp	Temperature.

In addition, the name of the material properties database file for each volumetric named component must be specified. The "no slip," "free_slip," "adiabatic," or "constant_temperature value" must be specified for each surface. The phase "liquid," "mush," "solid," "empty," or "always solid" must also be specified for each named component. "Always solid" means that there is no phase change and that the component is always solid. We emphasize that "always solid" is used for all named components in Electroconsolidation. In MaPS, the named components must be in capital letters and their corresponding conditions should follow until another named component, in uppercase letters, appears in the file; then, its conditions should be entered.

In the Section 9.5, we explain the *BC file, which can be created with the Vbounds module of MaPS or the vi editor in the Unix system.

9.5 *BC File Description

In MaPS, the geometry details are read via the neutral file, which contains some named components. The following may be added with the named components to the *BC file or may be created with the Vbounds module of MaPS to set up the initial and boundary conditions.

The specification records have a unique meaning in Electroconsolidation. These records are to be included in the *BC file, which can be created either with the vi editor or the Vbounds module of MaPS (the latter facilitates setting up the initial and boundary conditions). Below, an example *BC file is given as it is created by the Vbounds module. Based on this *BC file, explanations for the thermal structure assignments are made as follows:

```
================================================================

! CaPS-3D 2.0  Vbounds 14.1  03/14/00  16:29:33  grace
 CAPSGEOMETRY
  Al2O3                          /* property database file for material assignment */
  always_solid                   /* State of the material in this geometric section */
```

```
 temperature = 25.0
 vof       = 1.0
PREFORM
 Al2O3                          /* property database file for material assignment */
 always_solid                   /* State of the material in this geometric section */
 temperature = 25.0
 vof       = 1.0
POWDER-BT
 powder-bt1                     /* property database file for material assignment */
 always_solid                   /* State of the material in this geometric section */
 temperature = 25.0
 vof       = 1.0
POWDER-S
 powder-s                       /* property database file for material assignment */
 always_solid                   /* State of the material in this geometric section */
 temperature = 25.0
 vof       = 1.0
DIE
 die                            /* property database file for material assignment */
 always_solid                   /* State of the material in this geometric section */
 temperature = 25.0
 vof       = 1.0
RAM
 ram                            /* property database file for material assignment */
 always_solid                   /* State of the material in this geometric section */
 temperature = 25.0
 vof       = 1.0
FILCARB
 filcarb                        /* property database file for material assignment */
 always_solid                   /* State of the material in this geometric section */
 temperature = 25.0
 vof       = 1.0
FELT
 felt                           /* property database file for material assignment */
 always_solid                   /* State of the material in this geometric section */
 temperature = 25.0
 vof       = 1.0
LRAMBS
 wall
  noslip                        /* boundary conditions */
```

```
  temperature = 50.0                    /* Specified constant-temperature boundary */
URAMTS
 wall
  noslip                                /* boundary conditions */
  temperature = 50.0                    /* Specified constant-temperature boundary */
RAMS
 wall
  noslip                                /* boundary conditions */
  thermal_structure = 1                 /* transient heat-flux boundary */
FELTS
 wall
  noslip                                /* boundary conditions */
  thermal_structure = 2                 /* transient heat-flux boundary */
CENTER
 wall
  freeslip                              /* boundary conditions */
  adiabatic                             /* adiabatic heat-flux boundary */
THERMAL_STRUCTURE
! generic_ts                           /* 20 Characters long thermal structure variable */
! Generic Thermal Structure            /* 80 Characters long description of the structure */
 ts_face 3
 ts_area 1.0
 ts_temp 25.0
 htcoef 1
 n_mats 1
 ! constant_t
  n_part  2
  part_size  1.0
THERMAL_STRUCTURE
! generic_ts                           /* 20 Characters long thermal structure variable */
! Generic Thermal Structure            /* 80 Characters long description of the structure */
 ts_face 3
 ts_area 1.0
 ts_temp 25.0
 htcoef 2
 n_mats 1
 ! constant_t
  n_part  2
  part_size  1.0
HEAT_TRANSFER_COEFFICIENT
```

```
! generic_htcoef                    /* 20 Characters long coefficient variable */
! Generic Heat Transfer Coefficient  /* 80 Characters long description of the coefficient */
  htcoef_mul  1.0
  htcoef_len  6.35000E-03
  htcoef_rad  1.0
  htcoef_htc  1
HEAT_TRANSFER_COEFFICIENT
! generic_htcoef                    /* 20 Characters long coefficient variable */
! Generic Heat Transfer Coefficient  /* 80 Characters long description of the coefficient */
  htcoef_mul  1.0
  htcoef_len  6.35000E-03
  htcoef_rad  0.0
  htcoef_htc  1
HEAT_TRANSFER_CORRELATION
! generic_htc                       /* 20 Characters long correlation variable */
! Generic Heat Transfer Correlation  /* 80 Characters long description of the correlation */
  htc_c1  25.0
  htc_c2  .0
  htc_c3  .0
  htc_c4  .0

================================================================
```

In the above example, each line is explained by the /*...*/ comments. The named comments are "CAPSGEOMETRY," "PREFORM," "POWDER-BT," "POWDER-S," "DIE," "RAM," "FILCARB," and "FELT." The named sections "THERMAL_STRUCTURE," "HEAT_TRANSFER_COEFFICIENT," and " HEAT_TRANSFER_CORRELATION" have a special meaning and are used by MaPS to model thermal structures, and therefore cannot be used as named components for other parts of the geometry. Also, while using the Unix editor, such as vi, care should be taken in spelling the above names. The easiest way to set up the initializations for the thermal structures is to use the Vbounds load module of MaPS.

9.6 Thermal Structure Modeling

The purpose of implementing a thermal structure model is to permit consideration of heat-transfer interaction between the particulate medium and a solid in contact with the particulate medium in Electroconsolidation. The model solves one-dimensional heat conduction equations for all solid thermal structures; it calculates temperature distribution in solids and heat transfer from solids to surrounding air. In summary, the output from the thermal structure model is a heat source/sink term for the energy equation in Electroconsolidation.

To calculate heat transfer between air and solid surfaces (the die and the ram, in particular) in Electroconsolidation, a heat-transfer coefficient model may be required. A thermal structure is a solid structure in the air domain that is undergoing heat-transfer interaction with the surrounding air. In MaPS, structure/air heat transfer is computed as

$$Q = A * h_{eff} * (T_s - T_{air}),$$

Where A is the area, T_s is the temperature of the structure, T_{air} is the temperature of air, and $h_{eff} = (h_{conv} + h_{rad})$ is the effective heat transfer coefficient, which is the sum of both convection and radiation heat transfer coefficients. The heat transfer coefficients h_{conv}, h_{rad}, and h_{eff} of thermal structure "n," which are specified in the *BC input file or the Vbounds module, are defined by four input parameters:

ihtcor(n)	Convection heat transfer coefficient correlation number,
htclen(n)	Characteristic length in convection heat transfer.
htcmul(n)	Convection heat transfer coefficient multiplier.
htcrad(n)	Radiation heat transfer coefficient.

When the user-defined correlation number "ic" is

ihtcor(n) = 1 - 10, ic = ihtcor(n), and

Nu = c1(ic) + c2(ic) * Re**c3(ic) * Pr**c4(ic),

the convection heat transfer coeffici h_{conv} is defined as follows:

h_{conv} (n) = htcmul(n) * Nu * k / htclen(n).

When the user-defined correlation number "ic" is

ihtcor(n) = 11 - 20 and ic = ihtcor(n) - 10,

the convection heat transfer coefficient h_{conv} is defined as follows:

h_{conv}(n) = c1(ic) + c2(ic) * Re**c3(ic) * Pr**c4(ic).

Here,

k is the thermal conductivity of the ambient air,

Re = ρ*U*htclen(n)/μ, is the Reynolds number, where ρ is the density, U is the velocity, and μ is the viscosity of the material under consideration.

Pr = μ*C_p/k, is the Prandtl number, where C_p is the specific heat of the liquid, and

c1(ic), c2(ic), c3(ic), and c4(ic) are the Nusselt-number correlation coefficients.

The user can prescribe several correlations by entering various values of the coefficients c1, c2, etc. The Nusselt number and Reynolds number are based on the characteristic lengths of the system under consideration. These characteristic lengths are entered and must be prescribed by the user, because the Nusselt number Nu must always be positive. In Electroconsolidation, the ambient air (surrounding the die chamber) is assumed to be free flowing. Therefore, the Reynolds number (U = 0) is zero, the term "c2(ic) * Re**c3(ic) * Pr**c4(ic)" is always zero, and thus only c1(ic) is accounted for in the heat transfer calculation.

c1(ic) should be positive to accommodate a zero-flow situation (25.0),

c2 (ic) is a Nusselt-number coefficient (0.625),

c3 (ic) is a Nusselt-number coefficient (0.4),

c4 (ic) is a Nusselt-number coefficient (0.4).

The radiation heat transfer coefficient h_{rad} is defined as:

$$h_{rad}(n) = \sigma (T_s + T_{air}) (T_s^2 + T_{air}^2),$$

where σ is the Boltzman's constant, T_s is the temperature at the outside surface of the die chamber, and T_{air} (= 25°C) is the temperature of the ambient air that surrounds the die chamber.

A thermal structure is a collection of thermal-structure elements, each of which has the characteristics as specified by a thermal-structure prototype. Thermal-structure prototypes are defined by using the type, fluid, and material data lists in sequence. The order in which these are entered indicates the construction of the thermal structures and must conform to some specific rules as described in Section 9.6.1 below.

9.6.1 Thermal-Structure Data List

Correspondingly, the data list is referred to as the Thermal-Structure List in Vbounds. A THERMAL_STRUCTURE data list must begin the definition of each thermal_structure prototype. The initial default for all data list variables is zero. Subsequent defaults are the values in effect after reading the previous data list. The definition of thermal-structure prototype n + 1 must follow the definition of thermal-structure prototype n. The precise definition of each data list is given in the following:

ts_area	Surface area of the thermal structure. The absolute value is the surface area divided by the cell area. (Area Mutiplier [1.0])
ts_temp	Initial surface temperature of the thermal structure. (Temperature [25.0])
mats	Material to be specified (Air, constant_t.prop in Electroconsolidation).

n_part	Number of partitions in the material. A thermal-structure temperature will be computed for each material partition (Number of partitions). In Electro-consolidation, it is number of partitions in air.
part_size	Partition size, m, (Partition size).
htcoef	This number corresponds to an index or ordinal number and is an indicator of which of the available HEAT_TRANSFER_COEFFICIENTS are to be utilized in the analysis.

9.6.2 Heat Transfer Coefficient Data List

Correspondingly, HEAT_TRANSFER_COEFFICIENTS are referred to as the "Heat Transfer Coefficient List" in the Vbounds module. Referring to the *BC file given earlier, the various assignments under the name HEAT_TRANSFER_COEFFICIENTS include the first two lines as the comment lines: htcmul, the heat transfer coefficient multiplier (1.0); and htclen, the characteristic length (6.35E-03) as variables. Also, the Heat Transfer Coefficient Data List must call the correct heat transfer correlation variables via the index ihtcor.

9.6.3 Heat Transfer Correlation Data List

The Heat Transfer Correlation Data List,

c1 (c1) (25.0)

c2 (c2) (0.0)

c3 (c3) (0.0)

c4 (c4) (0.0),

completes the thermal-structure setup of Electroconsolidation modeling.

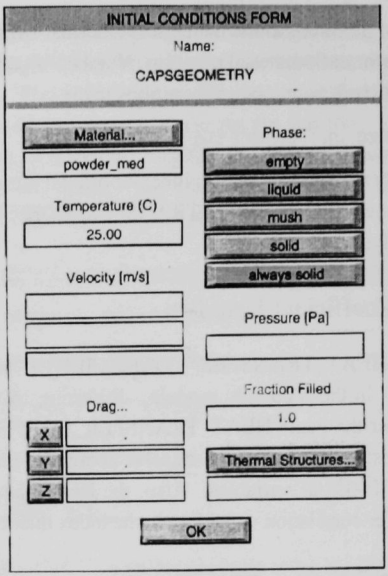


Fig. 3. Initial Conditions Form.

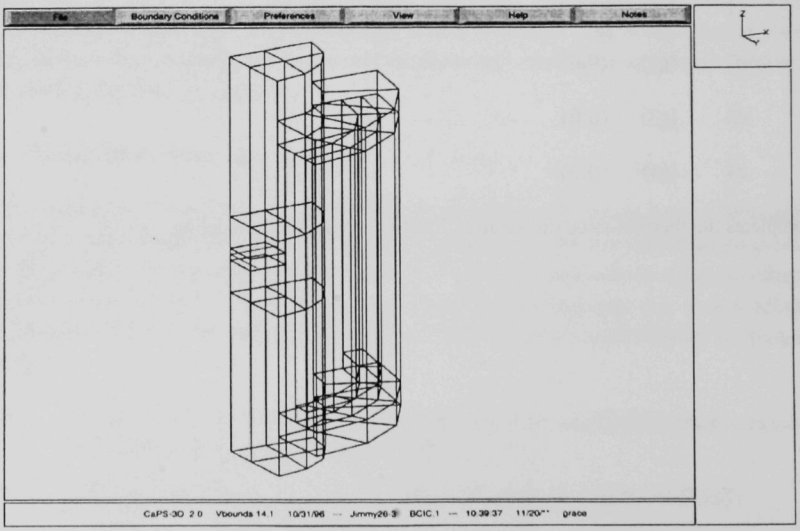


Fig. 4. Problem geometry displayed on screen according to neutral file created by CPRE.

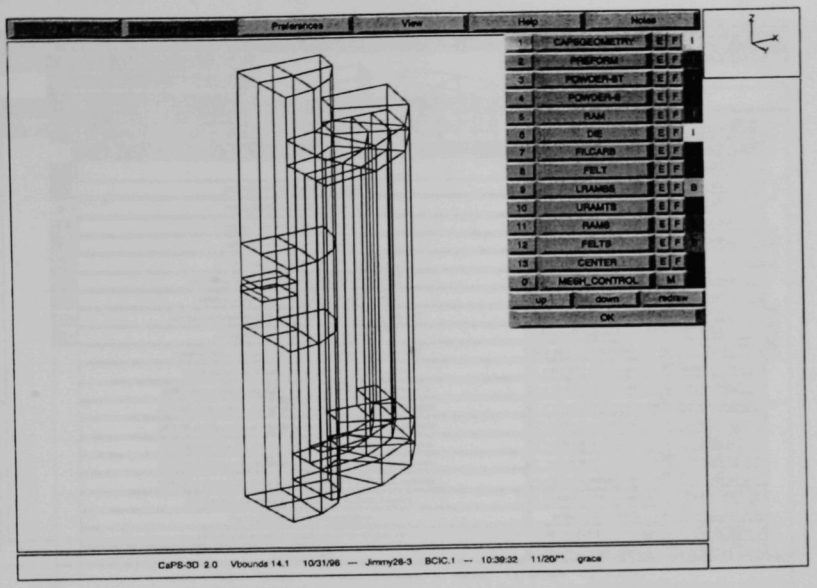


Fig. 5. List of named components that appear when "Boundary Conditions" button shown in Fig. 4 is depressed

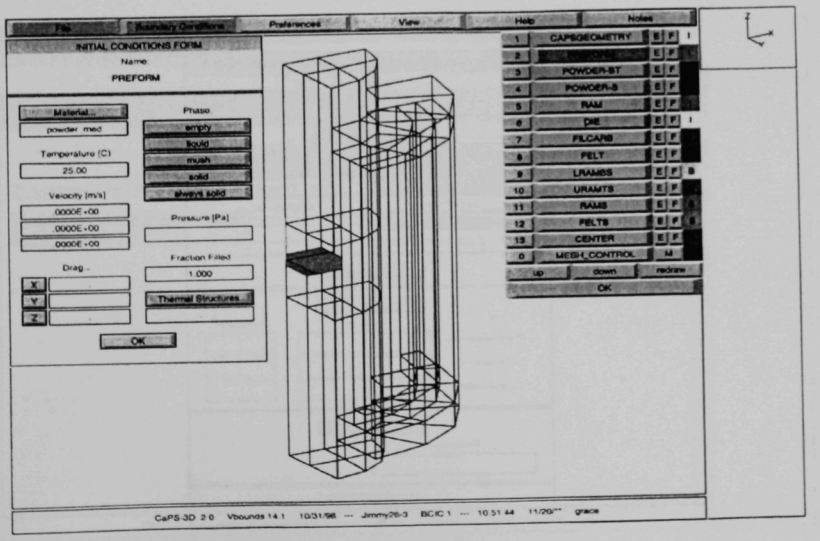


Fig. 6. Screen that appears when PREFORM button in Fig. 5 is selected.

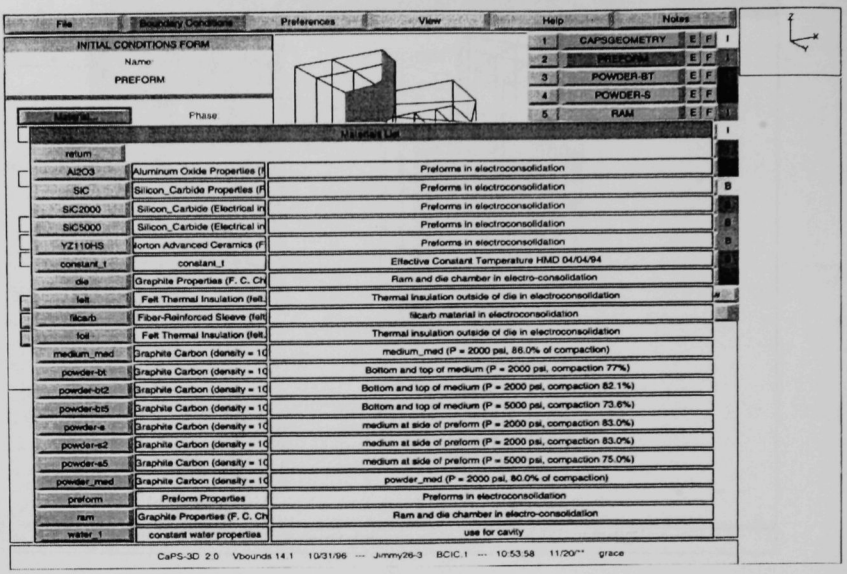


Fig. 7. Property data base file as called by MaPS: allows user to choose appropriate material for named component.

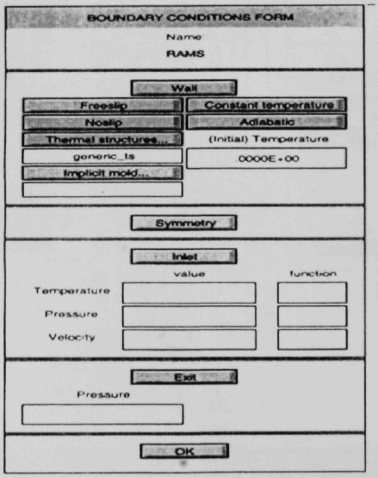


Fig. 8. Boundary Conditions Form.

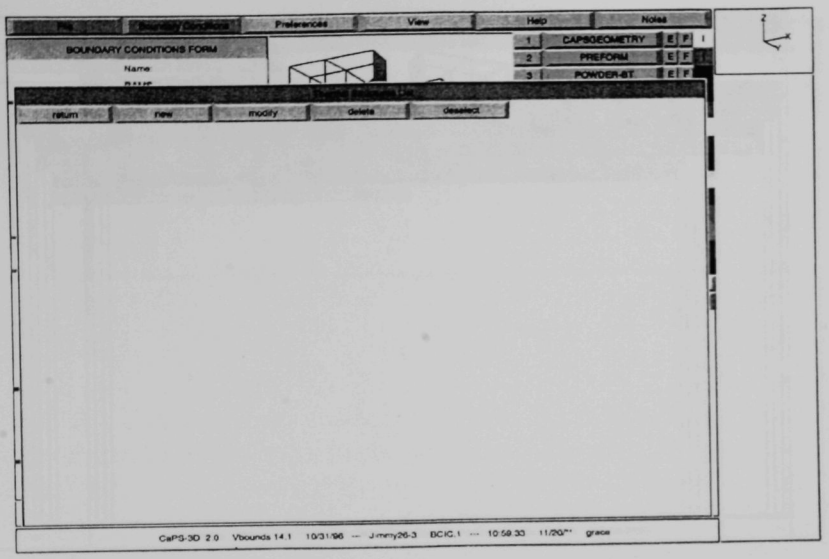


Fig. 9. Screen initialized when Thermal Structure... option is selected from Boundary Conditions Form.

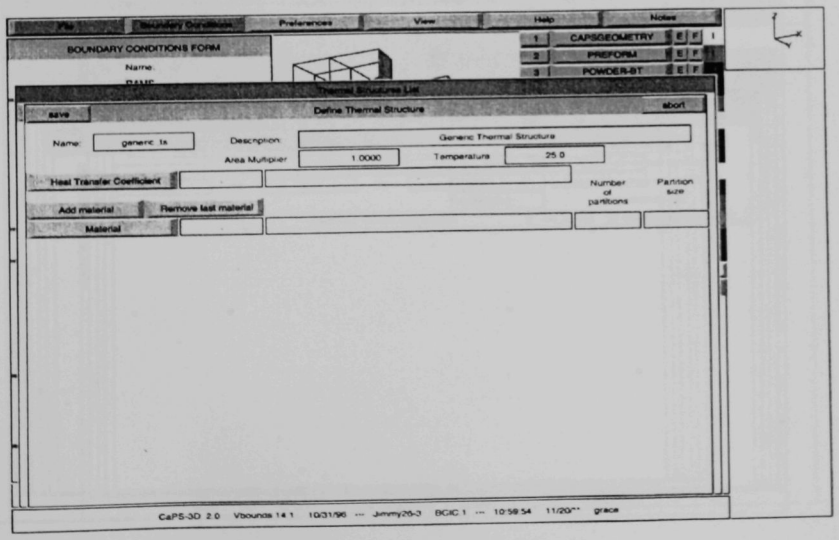


Fig. 10. Screen that appears when "new" option is selected in Fig. 9.

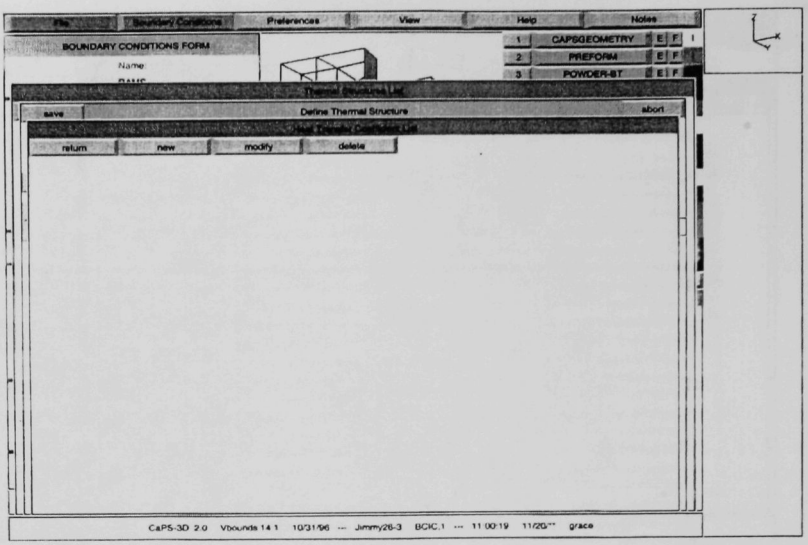


Fig. 11. Screen that appears when Heat Transfer Coefficient option is selected in Fig. 10.

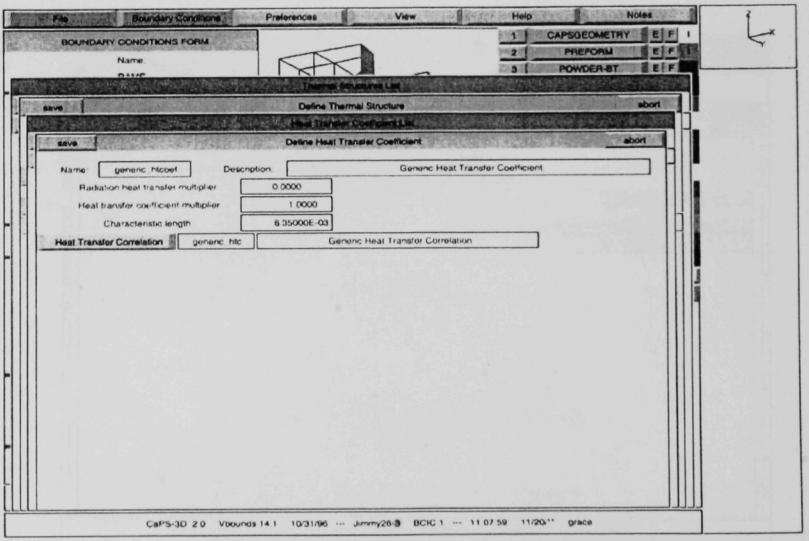


Fig. 12. Screen obtained when "new" option is selected in Fig. 11.

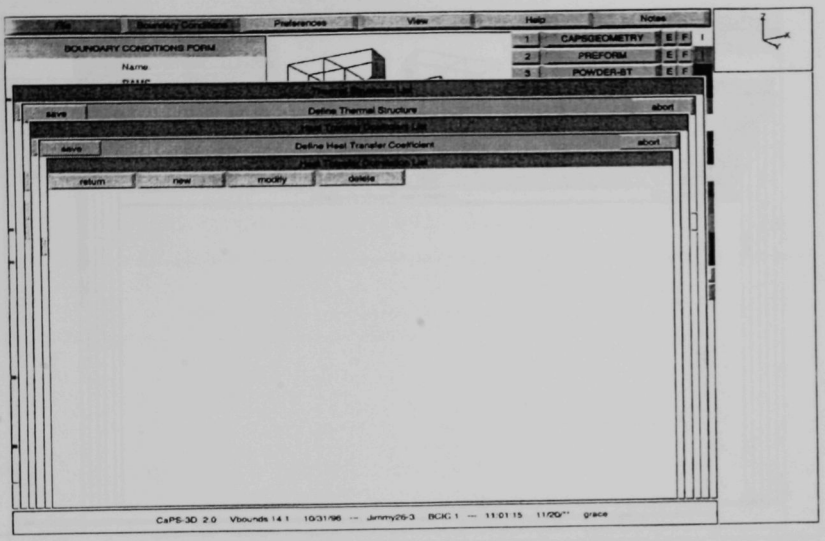


Fig. 13. Screen obtained when Heat Transfer Correlation is selected in Fig. 12.

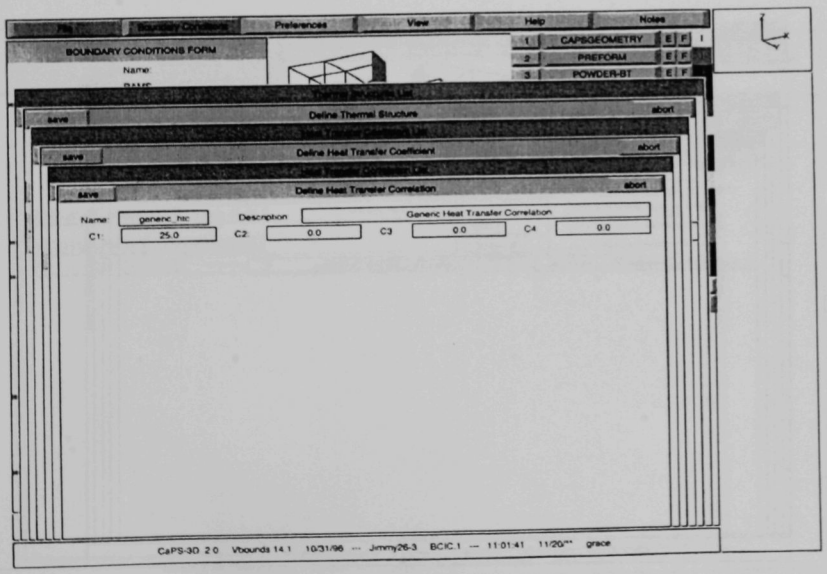


Fig. 14. Screen that appears when "new" is chosen in Fig. 13.

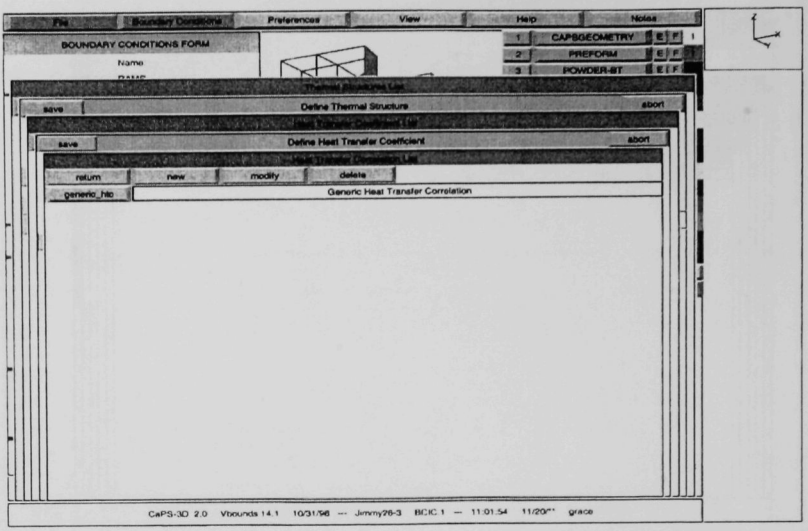


Fig. 15. Screen obtained when "save" is selected in Fig. 14.

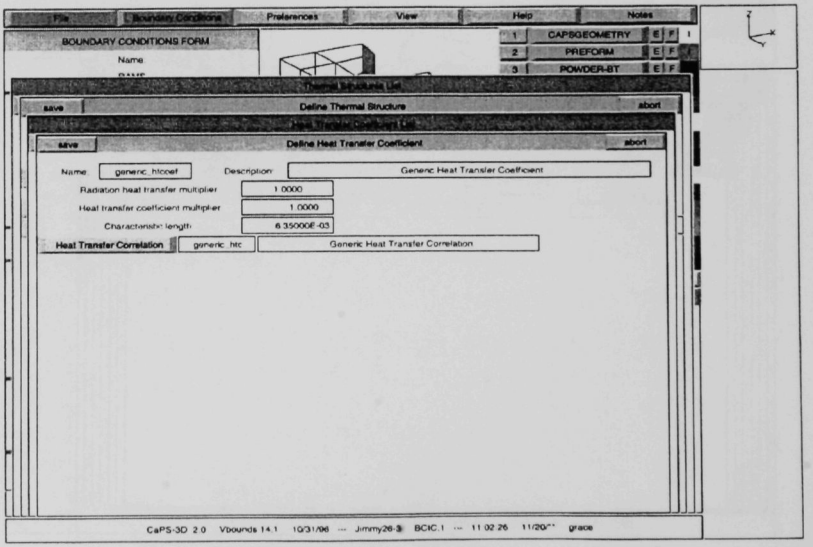


Fig. 16. Screen obtained when "return" is depressed in Fig. 15.

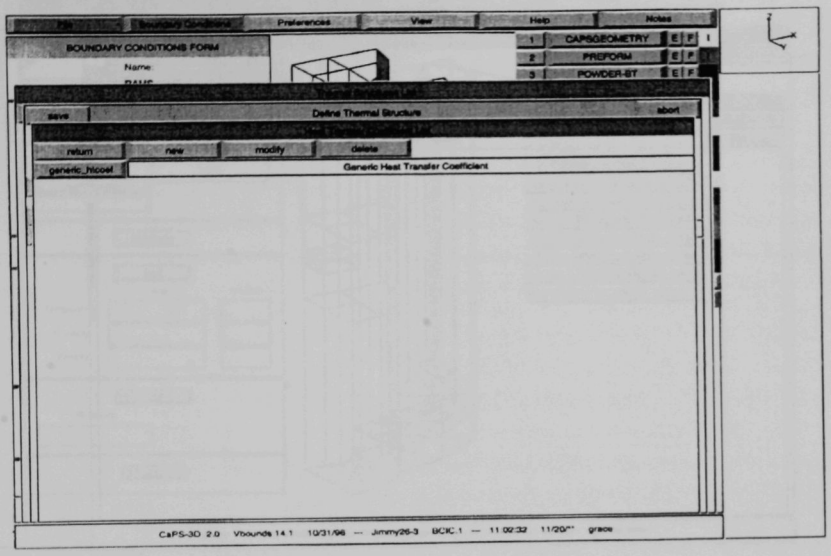


Fig. 17. Screen produced by selecting "save" in Fig. 16.

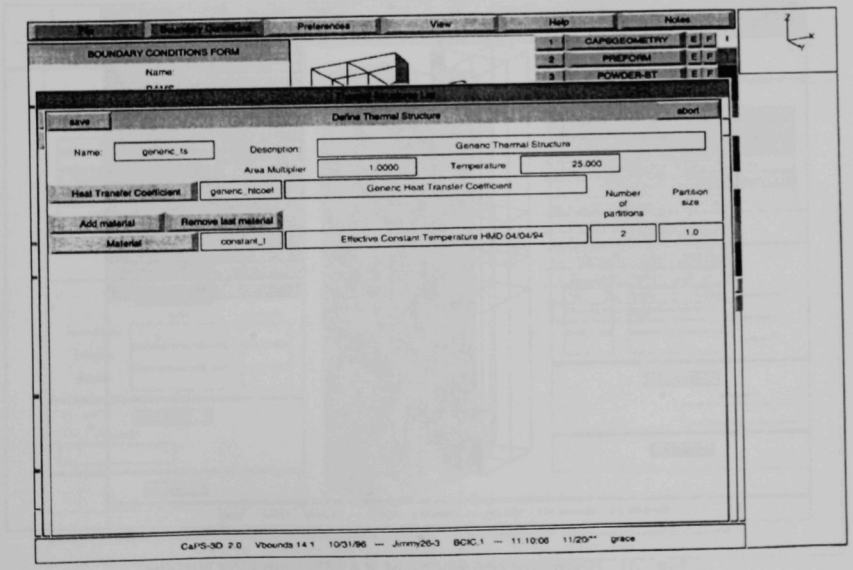


Fig. 18. Complete parametric listing that is saved.

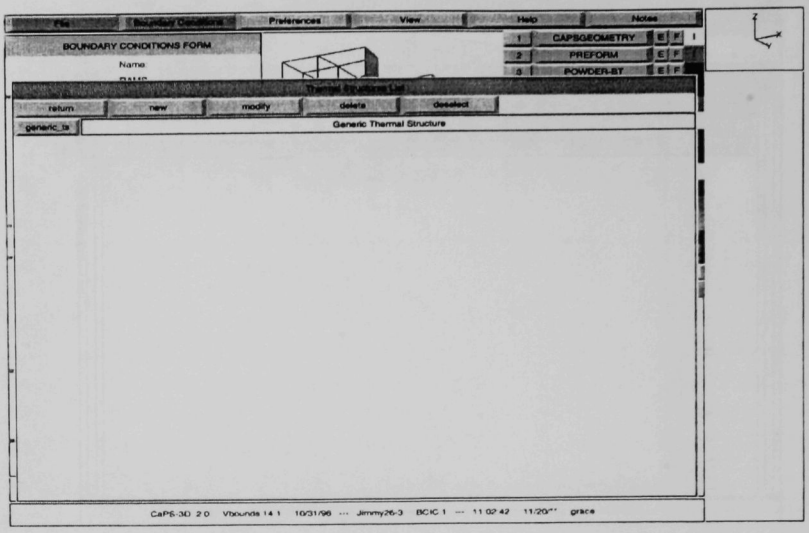


Fig. 19. Screen obtained by selecting "save" in Fig. 18.

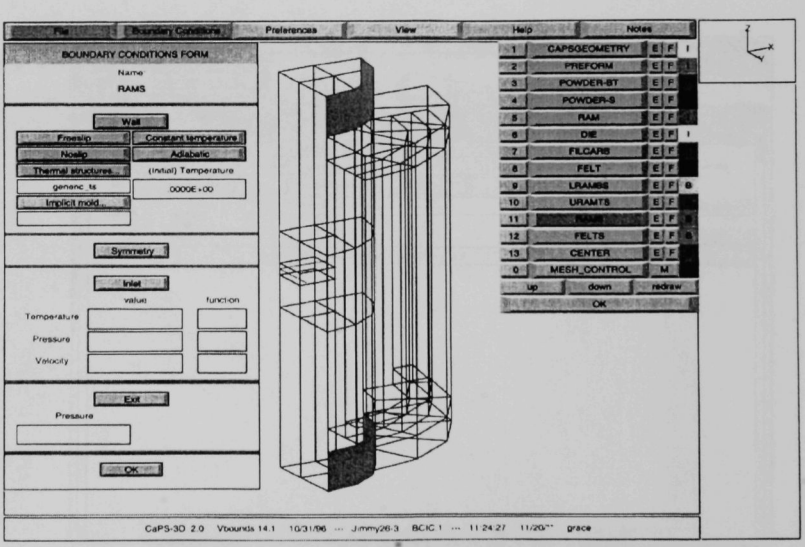


Fig. 20. Boundary conditions of RAMS (ram side surface), obtained by depressing "return" in Fig. 19.

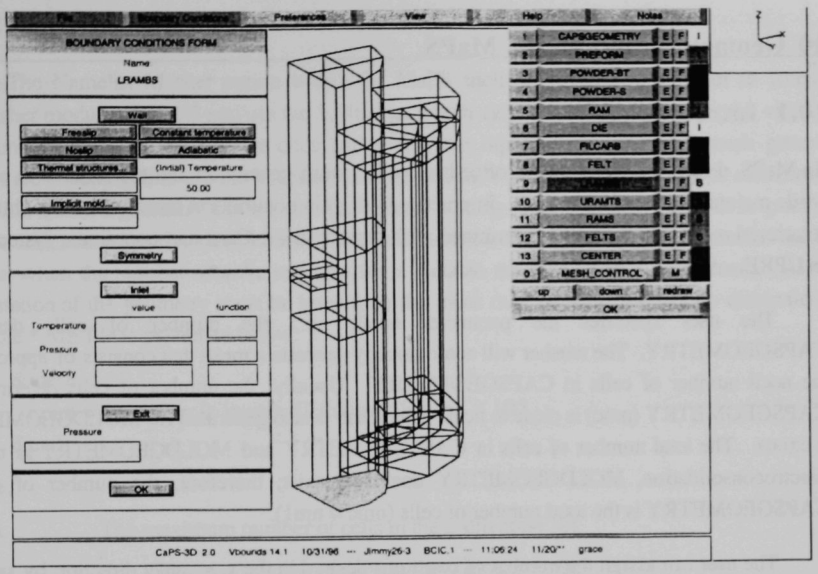


Fig. 21. Boundary conditions of LRAMBS (lower ram bottom surface).

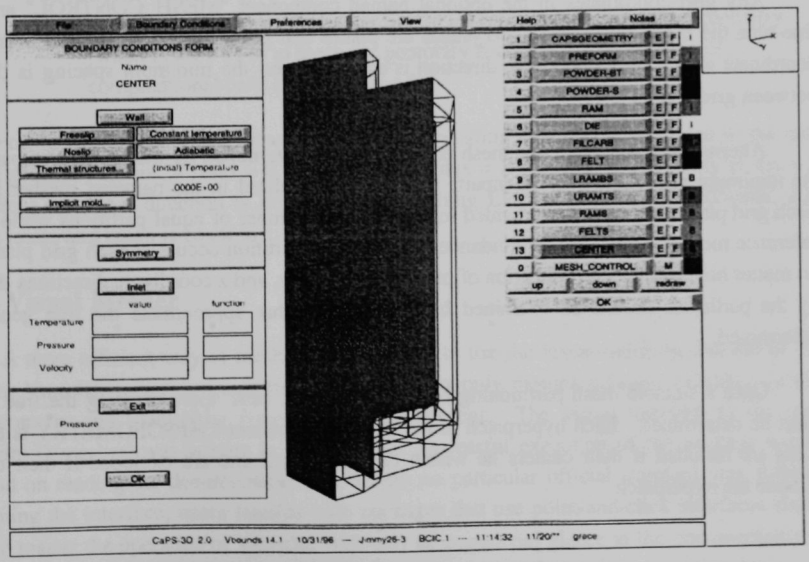


Fig. 22. Boundary conditions of CENTER (central surfaces of geometry setting).

10 Geometry Meshing of MaPS

10.1 Introduction

In MaPS, determination of the set of cells is called mesh generation and the process is performed while maintaining a uniform mesh. In practice, a slightly nonuniform mesh is usually obtained. A structured mesh partitioning is determined with respect to the Cartesian coordinate system utilized in CPRE.

The user specifies the parameter ncell, i.e., the number of cells desired in CAPSGEOMETRY. The mesher will automatically generate a mesh that consists of approximately the ncell number of cells in CAPSGEOMETRY. Usually, the number of cells determined for CAPSGEOMETRY (nmc) is close to ncell. The mesh is also generated for MOLDGEOMETRY, if it exists. The total number of cells in CAPSGEOMETRY and MOLDGEOMETRY is nml. In Electroconsolidation, MOLDGEOMETRY does not exist; therefore, the number of cells for CAPSGEOMETRY is the total number of cells (nmc = nml).

The user can assign the number of partitions desired in the x, y, and z direction by specifying inum, jnum, and knum. If all inum, jnum, and knum are specified, a mesh will be determined according to these three parameters and the parameter ncell is ignored.

Any grid coordinates in the optional named component "MESH_CONTROL" are used to fine-tune the mesh partitions. Grid planes are planes that pass through a grid and normal to the coordinate axes. Each coordinate direction is examined and the minimum spacing is determined between grid planes.

Alternatively, the reference mesh partition size is determined by the number of partitions of the minimum grid plane interval (npart) that can be initialized in the namelist mesher input file. Each grid plane interval is then divided into an integral number of equal partitions while using the reference mesh partition size for guidance. At least one partition occurs at each grid plane interval no matter how small. The collection of mesh planes in x, y, and z coordinate directions determined by the partitions should be examined before the cells that approximate the true geometry are determined.

Once a suitable mesh partitioning has been selected, cells approximating the true geometry must be determined. Each hyperpatch in the named component CAPSGEOMETRY is examined. Cells are included if their centers lie within the hyperpatch and are excluded if their centers lie outside the hyperpatch.

10.2 Namelist Mesher

The Namelist Mesher option within the MaPS shell scripts allows the user to forego the Vmesher module of MaPS and use the *.ME file, which can be created with either the vi editor or by running the Vmesher module once and saving the input upon successful mesh generation. Users are encouraged to use the Vmesher module of MaPS so the geometry can be confirmed and the final meshed geometry can be viewed. Thus, starting with a coarse mesh, it is possible to refine and, consequently, view the mesh until a suitable mesh is obtained. The namelist mesher is useful when the number of cells required for a suitable mesh is known beforehand. Also, the orientation of the geometry must be known so the mesh may be restricted in two dimensions. A sample *.ME file is presented below:

```
&mesh
  npart = 0, inum = 0, jnum = 0, knum = 1, ncell = 100
&end
```

Here,

imax	The maximum number of cells in the x-direction (0).
jmax	The maximum number of cells in the y-direction (0).
kmax	The maximum number of cells in the z-direction (1).
ncell	Total number of computational cells in the geometry.
npart	The flag to force meshing according to the two closest grids in the geometry (when npart = 1) or to mesh the geometry according to the number of computational cells (when npart = 0).

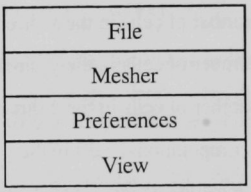
If any of the flags imax, jmax, kmax = 1, the corresponding direction is neglected in the meshing of the geometry. Thus, if imax = 0, jmax = 0, and kmax = 1, meshing is done only in the x and y directions and the z direction is neglected. In meshing Electroconsolidation problems, none of imax, jmax, and kmax is equal to 1.

10.3 Visual Mesher

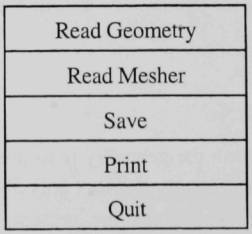
A more efficient way to mesh the geometry is to use the visual-meshing module of MaPS, i.e., the Vmesher, which is a user-friendly tool that not only meshes but also facilitates viewing of the geometry and identifying flaws, if any are present. The visual interface to the meshing program (Vmesher) was written in the hope that successful execution of the package would not depend on reading the documentation. Although no particular official standard was followed in designing the interface, users familiar with packages that use point-and-click interfaces should be able to master the operation of Vmesher with only an occasional glance at the documentation. The user is strongly encouraged to point-and-click first and read the documentation later. When

Vmesher is initiated, a window that is divided into four subwindows appears. In the largest viewing window, the hyperpatches in CPRE's named component CAPSGEOMETRY is displayed. A message window appears directly below the viewing window. Various instructional and informational messages will appear in this window. In the upper right-hand corner is a small window that shows the 3-D axis orientation diagram. Below the axis window is the command window, which always displays the top-level commands of the Top Level menu shown below. Selecting one of these commands leads to the appearance of the subcommands of the selected command below the top-level menu. By selecting the same command a second time, the submenu is removed. If a different top-level command is selected, the original submenu is removed and the submenu of the newly selected command is displayed.

The five command menus are the top-level menu, the file subcommand menu, the mesher subcommand menu, the preference subcommand menu, and the view subcommand menu.



Top-level menu



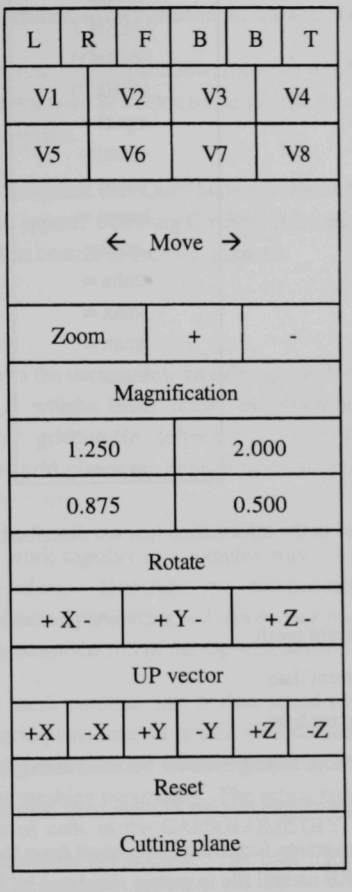
File subcommand menu

npart	0
X axis	0
Y axis	0
Z axis	0
ncell	1000
Preview	
Generate new mesh	
Read mesh	
View MaPS mesh	
View mold mesh	
Geometry	

Mesher subcommand menu

MaPS	
Edges	Faces
Mesh	Box
Mold	
Edges	Faces
Mesh	Box
No hidden	
Hardware Z-buffer	
Software Z-buffer	
Painters	
Spider-web	
Z-sort only	
No lighting	
Flat lighting	
Gouraud lighting	
Phong lighting	
Show info	
Triangles	

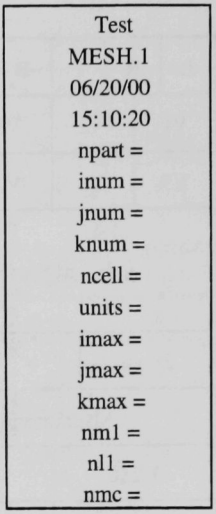
Preference subcommand menu



View subcommand menu

10.3.1 File Submenu

Show info. When information is available about an existing mesh or about the parameters used to define a new mesh, it may be displayed in a box below the file submenu by pressing the "Show info" button under the File menu. "Units" is the conversion factor used to convert the geometric neutral file to meters. Internally, MaPS works only in SI units. After successful mesh generation, for an example problem, the following information box was displayed:



The parameters displayed in the information box are described below.

TEST	
MESH.1	Current mesh
06/20/00	Current date
15:10:20	Current time
npart	Number of partitions for the minimum grid plane interval
ncell	Approximate number of mesh cells desired in the MaPS geometry
units	Conversion factor used to convert from the original units used in the CPRE neutral file to meters requested by MaPS
imax	Actual number of cells in the x direction
jmax	Actual number of cells in the y direction
kmax	Actual number of cells in the z direction
nm1	Total number of cells in the CAPSGEOMETRY and MOLDGEOMETRY
nl1	Number of surface elements (exposed faces) in the meshed configuration
nmc	Total number of cells in the CAPSGEOMETRY

Read Geometry: Read in all hyperpatches in CPRE named component CAPSGEOMETRY.

Read Mesh: Read in the mesh cells created previously.

Save: When the Save button is pressed, a file that contains the Mesher input parameters is written.

Print: A postscript file of the current viewing window, message window, axis window, and, if visible, the meshing information window, is written to the file file.ps and printed to the default printer when the Print button is pressed.

Quit: To terminate the Vmesher program, the "Quit" button is selected. This in turn causes Cancel Quit and Confirm Quit buttons to appear. Selecting Confirm Quit causes Vmesher to terminate, while selecting Cancel Quit returns control to the File submenu.

10.3.2 Mesher Submenu

The Mesher submenu allows the user to view an existing mesh or enter the specifications for a new mesh and "Preview," which is to draw an all-encompassing meshbox around CAPSGEOMETRY and any gridpoints defined in the CPRE named component MESH_CONTROL. Additional grid planes are defined by considering the five input parameters described below.

The five input parameters work together in a complex way. A short explanation alluding to their general usage is given below. However, we recommend that the user try several combinations of the mesh specification parameters and observe the resultant mesh boxes, using the "Preview" button to clearly understand the roll of the input parameters.

The f reference mesh partition size is determined by dividing the minimum grid plane interval by npart. Each grid-plane interval is then divided into an integral number of equal partitions by using the reference mesh partition size for guidance. The next four parameters give the user-requested values for the meshing parameters. The actual number of cells in the x, y, and z directions and the total number of cells in the CAPSGEOMETRY will be computed with these input values and additional constraints considered. The final mesh values may not match the requested values.

x-axis	The requested number of partitions in the x-direction.
y-axis	The requested number of partitions in the y-direction.
z-axis	The requested number of partitions in the z-direction.
ncell	The requested number of mesh cells in the CAPSGEOMETRY.

Preview. Selecting the preview button draws a rectangular solid around CAPSGEOMETRY and overlays the mesh lines on this meshbox.

Generating a new mesh. The "Generate new mesh" button runs a background job to generate files that contain the mesh information needed by MaPS. The cells in this mesh will be the cells drawn by Preview, which are inside CAPSGEOMETRY. This process may take anywhere from a few seconds to many minutes of CPU time, depending on the number of cells in the final mesh. While the process is in progress, the message "Generating a new mesh" appears in the message window and execution of Vmesher is suspended. Upon successful completion of the mesh generation, the message window is cleared.

Read Mesh. The "Read Mesh" button reads the mesh cells just generated in CAPSGEOMETRY.

View CaPS mesh. The "View CaPS mesh" toggles the CAPSGEOMETRY mesh on and off.

View mold mesh. The "View mold mesh" toggles the MOLDGEOMETRY mesh on and off. MOLDGEOMETRY mesh does not exist in Electroconsolidation analysis.

Read Geometry. Read in all hyperpatches in CPRE named component MAPSGEOMETRY.

10.3.3 Preference Submenu

This submenu allows the user to set various preference parameters.

CaPS Edges	The edges of the MaPS geometry are drawn.
CaPS Faces	The faces of the MaPS geometry are represented by a net of surfaces.
CaPS Mesh	The mesh of the MaPS geometry is drawn.
CaPS Box	A box of rectangles is drawn on the MaPS geometry.
Mold Edges	The edges of the mold geometry are drawn (not available in Electroconsolidation).
Mold Faces	The faces of the mold geometry are represented by a net of surfaces (not available in Electroconsolidation).
Mold Mesh	The mesh of the mold geometry is drawn (not available in Electroconsolidation).
Mold Box	A box of rectangles is drawn on the mold geometry (not available in Electroconsolidation).
No Hidden	No hidden-surface calculation is made before rendering hyperpatch faces or meshed cells.
Hardware Z-Buffer	When the hidden surfaces are used for hyperpatch faces or meshed cells, hardware will be used if available.

Software Z-Buffer	This option skips over the hardware and may display fewer edge effects than a hardware Z-buffer, depending on the hardware used.
Painters	This option skips over the hardware and may show fewer edge effects than a hardware Z-buffer.
Z-Sort Only	When the Z-Sort Only button is selected, a simple sort of geometry by approximate screen depth will be sufficient to generate an accurate picture.
No Lighting	The faces of a shell or a mesh are to be drawn as really curved surfaces.
Flat Lighting	The faces of the geometry are to be drawn as flat objects.

10.3.4 View Submenu

The View submenu allows the user to change the orientations of the image in the viewing window by rotating or moving the object, by changing the viewpoint, by magnifying the object, or by zooming into the window. A right-handed coordinate system is used, with the x axis pointing to the right, the y axis pointing upward, and the z axis pointing out of the screen.

Left	(L)	A view of the left side of the image is presented.
Right	(R)	A view of the right side of the image is presented.
Front	(F)	A front view of the image is presented.
Back	(B)	A back view of the image is presented.
Top	(T)	A top view of the image is presented.
Bottom	(B)	A bottom view of the image is presented.
Views		The View buttons allow the user to change the viewpoint of the viewer or to orbit the contents of the viewing window.
Move		This selection allows moving the geometry with respect to the screen. The Move button in the View submenu provides for moving the image in the view window in the direction indicated by the arrows on the button a fixed distance each time the button is selected.
Zoom		When the Zoom button is selected, a message appears that instructs the user to define a rectangular region by pressing the mouse button down on one corner and releasing it on the other corner. While the mouse button is pressed and the pointer is moved, the user can select a box size that delimits the resultant rectangle.

Magnification The Magnification button in the View submenu provides the opportunity to enlarge or reduce the image, based on the ratios (1.250, 2.000, 0.875, or 0.5000) in the viewing window.

Rotate The Rotate button in the View submenu provides the means to rotate around the x, y, and z axis a fixed number of degrees each time the button is selected. The Rotate button is partitioned into a counterclockwise and clockwise region for each axis.

Reset The image in the viewing window is reset to the default front view.

11 Input and Output Data of MaPS

11.1 *TS Input File Description

11.1.1 Introduction

When the geometry is set, the initial and boundary conditions for the various sections of the geometry are assigned, and the geometry is meshed; a simulation run can be conducted. The simulation requires input parameters that define the extent of the run, time-step size, etc. This input file, the *TS file, is created with the vi editor. The various run-control parameters are described below. Only a few parameters are required but all of the parameters are included to demonstrate the vast applications of the MaPS software. The computational domain is partitioned into several computational cells, each bounded by consecutive x- and y-direction grid planes. Surfaces (portions of a plane) may be defined on the exterior, bounding the computational domain, and in the interior. The intersection of a surface and consecutive grid planes outlines a surface element.

User input is read from a file associated with Unit 5 and is called the *TS file, where the * signifies the project name. This input consists of two required name lists. The *BC file, which may contain several other record groups (such as thermal structures), also facilitates in setting up the initial and boundary conditions. The term record corresponds to the earlier concept of a card image. The user may specify Unit 5 or the *TS input file in any order. One possible order is

Problem description and comments (Optional)
Namelist/geom/
Namelist/data/

Below, we describe the variables that may be used in the Namelist/geom/ and Namelist/data/ sections of the TS file. An asterisk in parenthesis (*) indicates that the corresponding value is assigned to the variable as default.

11.1.2 Namelist/geom/

iemf		Flag to include electromagnetic field calculations
	= 0	No calculations of electromagnetic fields
	= 1	Calculations of electromagnetic fields
ifvof		Flag to enable volume of fluid calculations
	= 0	No calculations of volume of fluid
	= 1	Calculations of volume of fluid
isolve		Flag to determine the solution technique used to solve the energy equations

= 0 Successive-overrelaxation (SOR) solution scheme is used

11.1.3 Namelist/data/

alpha
= 0.0 Semi-implicit time advancement for energy equations
= 1.0 Fully implicit time advancement for energy equations (*)

ifener
= 0 No energy calculation
= 1 Energy calculation is performed (*)

istate
= 2 Beginning of a transient run (*)
= 3 Continuation of a transient run

Time- and Time-Step-Related Parameters. These parameters are

dt(1) Initial time-step size in seconds

dt(2) Maximum time-step size, in seconds. This value is used when automatic time step is enabled, idtime = 1.

idtime
= 0 Time-step size is taken from the user-specified variable dt.
= 1 Time-step size is computed internally as the product of the largest allowable time increment, given the conditions (Courant time-step size and a user-specified variable, rdtime).

ntmax Maximum time-step number for this run. Normal termination occurs after completion of this time step (99999).

nwrite Number of steps to write selected output data in SELECT.dat file

rdtime Time-step size is computed internally as the product of the largest allowable time increment, given the conditions. This value is used only if idtime = 1.

timax Maximum time of this run in seconds (3.6E+7). Normal termination occurs after this time has been reached. timax refers to the simulation or problem time, not the computer CPU time needed to run the problem.

tstart Initial time in seconds (0.0). This value should be reset to zero at the beginning of a transient run, istate = 2.

ntplot Up to 25 values to specify when step information is to be written. The following are acceptable values of ntplot:

= 0 No more step information is written (*).

> 0 Time-step number for which step information is written. After the nth

positive time step in ntplot has been processed, the (n+1)th value of ntplot is used to determine which subsequent time steps are written to the RUN directory.

< 0 A value -n indicates that information is written to the RUN directory every nth time step. No subsequent values of ntplot are considered. Example: ntplot = -5 indicates that every fifth step is to be processed. ntplot = 5, 10 -20 indicates that steps 5, 10, 20, 40, 60, etc., are to be processed. ntplot = 10, 20, 0 indicates that only steps 10 and 20 are to be processed.

Iteration Control Parameters. The iteration control parameters are

eps5	Energy solver convergence criterion parameter (1.0E-5)
it(1)	Number of iterations permitted for time steps
itmaxe	Number of iterations in the energy solver (99)
omegae	Underrelaxation factor for the energy equation coefficients (1.0)
relaxe	Relaxation factor for the SOR energy solver (0.95)

Geometry and Operation Control Parameters. The geometry and operation control parameters are

icase	Electroconsolidation flag
= 1	Double-action pressing of Electroconsolidation (*), without electrical insulation between medium bed and die
= 2	Double-action pressing of Electroconsolidation, with electrical insulation between medium bed and die
compjc	Janssen constant (0.0).
dofdie	Outside diameter of die chamber (0.0)
dofmed	Inside diameter of die chamber (0.0)
douter	Outside diameter of fiber-reinforced sleeve or thermal-insulation felt (0.0)
emiss	surface emissivity for radiation heat transfer (1.0)
findex	Friction index of compaction (0.0)
hofdie	Height of die chamber (0.0)
hofmed	Height of particulate medium (0.0)
power	Power used at the ends of rams in ELEKTRA boundary conditions (0.0)

pram0 Applied pressure from the rams in Electroconsolidation (0.0)

sigma Stefan-Boltzmann constant [5.67e-8 J/(s-m^2-k^4)]

Power-Supply-Curve Control Parameters: The power curves are modeled as a function of time in Electroconsolidation (Fig. 23). The coefficients and end times for the power curves are the input data in the *.TS file.

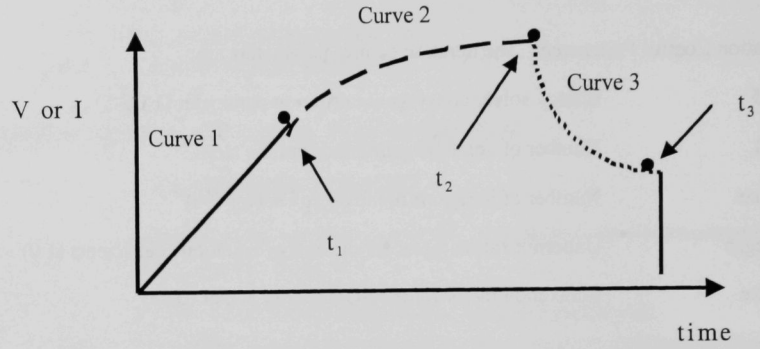


Fig. 23. Power curve as a function of time during Electroconsolidation

In Fig. 23,

Curve 1: V or I = $F_1(t) = a_0 t$

Curve 2: V or I = $F_2(t) = b_0 + b_1 t + b_2 t^2 + b_3 t^3 + b_4 t^4$

Curve 3: V or I = $F_3(t) = f_0 + f_1 t + f_2 t^2 + f_3 t^3 + f_4 t^4$

cofemf Coefficient a_0 for Curve 1 of power input (0.0)

bmod0 Coefficient b_0 for Curve 2 of power input (0.0)

bmod1 Coefficient b_1 for Curve 2 of power input (0.0)

bmod2 Coefficient b_2 for Curve 2 of power input (0.0)

bmod3 Coefficient b_3 for Curve 2 of power input (0.0)

bmod4 Coefficient b_4 for Curve 2 of power input (0.0)

fmod0 Coefficient f_0 for Curve 3 of power input (0.0)

fmod1	Coefficient f_1 for Curve 3 of power input (0.0)
fmod2	Coefficient f_2 for Curve 3 of power input (0.0)
fmod3	Coefficient f_3 for Curve 3 of power input (0.0)
fmod4	Coefficient f_4 for Curve 3 of power input (0.0)
clock1	End time (t_1) of Curve 1 of power input (3600e+5)
clock2	End time (t_2) of Curve 2 of power input (3600e+5)
clock3	End time (t_3) of Curve 3 of power input (3600e+5)

11.1.4 Sample of *TS Input File

```
------------------------------------------------------------------------------------------------------------
Six preforms in medium (test.TS.1)
------------------------------------------------------------------------------------------------------------

&geom
 ifvof=0,                          /* flag to enable volume-of-fluid calculation
&end                                  ifvof=0 means no fluid calculation */


&data
 itimer=1,                         /*
 istate=2,                         /* istate=2, beginning of a transient run */
 dt=0.25,                          /* time-step size */

 ntmax=7000,                       /* maximum time-step number for this run */
 ihtcor(1)=11,                     /* heat transfer coefficient correlation number
                                      of thermal structure 1 */
 ihtcor(2)=11,                     /* heat transfer coefficient correlation number
                                      of thermal structure 2 */
 icase=2,                          /* single- or double-action pressing */
 pram0=5000.0,                     /* applied pressure from rams */
 power=4.0,                        /* power used in ELEKTRA BCs */
 dofmed=0.127,                     /* inside diameter of die chamber */
 hofmed=0.1143,                    /* height of particulate medium */
 dofdie=0.127,                     /* outside diameter of die chamber */
 hofdie=0.2159,                    /* height of die chamber */
```

```
douter=0.2032,                    /* outside diameter of fiber-reinforced sleeve
                                     or thermal-insulation felt */

bmod0=0.38353,                    /* coefficient #1 for the 2nd curve of power input */
bmod1=9.7546e-3,                  /* coefficient #2 for the 2nd curve of power input */
bmod2=-1.1026e-5,                 /* coefficient #3 for the 2nd curve of power input */
bmod3=7.1314e-9,                  /* coefficient #4 for the 2nd curve of power input */
bmod4=-1.7217e-12,                /* coefficient #5 for the 2nd curve of power input */

nwrite=140,                       /* number of steps written in SELECT.dat file */
clock1=0.0,                       /* end time of the 1st curve of power input */
clock2=1500.0,                    /* end time of the 2nd curve of power input */
clock3=1500.0,                    /* end time of the 3rd curve of power input */

emiss=1.0,                        /* surface emissivity for radiation heat transfer

trest=75000.0,                    /* time allowed to run the computer for this job */
ntprnt= 3000,6000, -9999,         /* time-step numbers to call output data */
ntplot= 3000,6000, -9999,         /* time-step numbers to plot output data */
nthpr = 621012, 623018,           /* numbers of coded values to be printed */
&end

SELECT tl (  0.033, -0.01905,  0.01905)   /* locations of thermocouples to record temperature
SELECT tl (   0.0,   0.0381,  0.01905)       history in SELECT.dat file */
SELECT tl (  0.033,  0.01905, -0.01905)
SELECT tl (  0.001,   0.001,    0.0)
SELECT tl ( 0.02888,    0.0,    0.0)
SELECT tl ( 0.04838,    0.0,    0.0)
SELECT tl (  0.001,   0.001,   0.02)
SELECT tl (  0.001,   0.001,   0.04)
```

11.2 Print Step

11.2.1 Introduction

A hard-copy output of the various variables can be obtained by creating a *.PS file in the current working-run directory. This file informs MaPS which variables are to be printed and can

be initiated after any of the evolved step files, which are the binary simulation result files. As an example, this file appears as

K - plane 3
Variables ul p tl bemf cemf femf

Where the plane of interest is the kth-plane, clipped three cells in the z direction. Multiple planes can also be printed by writing the plane of interest in the *.PS file.

11.2.2 Description of Cell Output Variables

hl	Enthalpy (J/kg)
tl	Temperature (Celsius)
rl	Density (kg/m^3)
P	Pressure (Pa)
qsour	Volumetric heat source (w/m^3)
rcon	Molecular conductivity [W/(m*K)]
vfluid	Cell volume (m^3)
spheat	Specific heat [J/(kg*K)]
qemf	Resistivity heat (J/m^3)

12 Material Properties of MaPS

12.1 Introduction

Material properties are permitted to be temperature dependent in MaPS. Four material-property variables are required: thermal conductivity, density, specific heat, and viscosity. If temperature-dependent properties are available, MaPS linearly interpolates the adjacent properties to provide properties over the entire temperature range, and uses a more elaborate table. Beyond the last property value, MaPS assumes a constant equal to the last property value.

The advantage of using temperature-dependent properties rather than constants is that by doing so, the actual phase change at a particular temperature can be realized. Also, the temperature dependence corresponds to real-world properties, because properties vary as a function of temperature. If only constant properties are available for the particular material, e.g., the solid and liquid state properties, MaPS linearly interpolates the end points and assumes constant values beyond the table entries. In Chapter 12, we explain how to create a *.prop file in the properties subdirectory.

12.2 Property Database Organization

The properties directory contains the files that define the properties for all of the materials that are to be available to the MaPS software. The MaPS materials are organized in a library directory, where there exists a file called "available_materials," which is a table of contents of the materials in the library. The properties of a material are in files named "*.prop." The property data are in tabular format, and property data file names are listed in the file. The properties directory already contains several sample properties files. The file name must be no more than 11 alphanumeric characters long, and including the ".prop" extension, it must not be longer than 16 alphanumeric characters.

As mentioned earlier, properties are evaluated by linear interpolation of two adjacent tabular data at corresponding temperatures. The property value beyond the one that is last defined is considered a constant, with a value equal to the last-defined value. Each material file must contain tables that list the information shown in Table 7.

Table 7. Information to be listed in tables in material file

Property	Name	Units
Thermal Conductivity	KXX	(heat*length)/(time*area*temp)
Mass Density	DENS	mass/volume
Specific Heat	C	heat/(mass*temp)
Viscosity	VISC	(force*time)/length^2

The following is a sample file for the preform Aluminum-Oxide Al203. This file will be used to explain the various terms. Other materials properties used in Electroconsolidation are listed in Appendix C.

```
---------------------------------------------------------------------------------

/COM  Aluminum Oxide Properties (Al203.prop)
/COM  Preforms in electroconsolidation
MPTEMP
MPTEMP,  1,              20.0, 500.0, 1000.0, 1200.0, 1400.0,
MPDATA,KXX , 1,  1,  33.0,  11.4,     7.2,     6.7,     6.3,
MPTEMP
MPTEMP,  1,              25.0,
MPDATA,DENS,  1,  1, 3900.0,
MPTEMP
MPTEMP,  1,           20.0,   500.0, 1000.0, 1200.0, 1400.0, 1500.0,
MPDATA,C   ,  1,  1, 755.0, 1165.0, 1255.0, 1285.0, 1315.0, 1330.0,

---------------------------------------------------------------------------------
```

The following aspects of the above files must be noted:

/COM — Comment lines used to either write references, citations, or to_si conversions. The first two lines are read by the Vbounds module of MaPS, which assigns initial and boundary conditions to various parts of the geometry. Thus, the first /COM line must contain the name of the material, and the following line may contain a brief description of the material.

MPTEMP — These are the temperature-specified lines, which may contain, at most, five temperature values per line. The first four spaces after "MPTEMP" are to be used for the index of the following temperature value. Thus, " 1," implies

that the temperature value following the comma is the first temperature value, " 6," implies that the following temperature value is the sixth value, and so on.

MPDATA This is the prompt to define material property values. Following "MPDATA," either of the properties may be specified, limiting the variable name to four characters. Thus, KXX is the thermal conductivity, DENS is the density, C is the specific heat, and VISC is the viscosity. If the variable name is less than four characters, blank spaces must be specified until four spaces are utilized, followed by a comma. Thus, following "MPDATA,", we can have "KXX ,", "DENS,", "VISC,", or "C ,". The following three spaces are reserved for indexing, which is followed by a comma. The next three spaces indicate the index of the first data value on that line. Thus " 1," indicates that the first data value on that line is the first in the series, " 6," indicates that it is sixth in the series, and so on.

Note that the "to_si" unit conversions can be embedded within each definition of the material properties. In the above examples, the units are all SI and therefore, "to_si" parameters need not be used.

Thus, new materials may be made available to MaPS merely by adding a new property file into the materials property database file. After a new material file has been created, the "available materials" file must include the name of the new file so MaPS can identify it. This can be done by executing the script "make table," which searches for all of the *.prop files and appends their names into the available materials file.

13 MaPS Shell Scripts

13.1 Introduction

The MaPS shell scripts allow the user to conduct the simulation run by first creating the geometry, assigning the boundary or initial conditions, meshing the geometry that has been constructed, submitting the simulation runs, and analyzing the results via post-processing. The scripts are very user friendly.

Once MaPS is invoked, if a PROJECTS directory has not been previously defined in the current directory, the user is asked if one is needed. The user is then prompted to select a particular project name. Here, project refers to a particular geometrical definition. On this level, the current directory contains a directory named PROJECTS. Within the PROJECTS directory are the individual subprojects.

A particular subproject can have several meshing alternatives and several alternative boundary and initial condition specifications. The alternative meshes are named sequentially as MESH.1, MESH.2, etc. Similarly, alternative boundary and initial condition specifications are named sequentially as BCIC.1, BCIC.2, etc. A model is defined to have a particular mesh as well as a particular boundary and initial specification associated with it. The various subproject models are named sequentially MODEL.1, MODEL.2, etc. For example, MODEL.i may have MESH.j and BCIC.k associated with it. In this way, various subproject models can have the same boundary and initial condition specifications with differing meshing alternatives. The mesh is the rendering of the project geometry based on user specifications. Boundary and initial-condition specification is done with respect to the subproject geometry and is therefore independent of mesh.

Default selections based on "carriage return" allow the user to move quickly through the simulation. Also, the defaults for the file path can be changed while in scripts, and the Bcgen, Mesh, RUN jobs are submitted in the background. To create the geometry, the scripts link to the CPRE geometry modeler. Vmesher, the visual meshing module, and Vbounds, the visual initial and boundary conditions setup module, can be activated with the MaPS shell scripts. If the CELLDATA, MAP, etc., do not exist after geometry meshing, the scripts remind the user to look into the setup and resubmit the mesh job. The same is true for assigning the input for BCIC. If several neutral, mesh, *BC, *TS, or input files exist in the working directory, the user can choose the required file for the particular model, mesh, BCIC, and run, by using the scripts. Furthermore, the scripts are linked so that postprocessing of the simulation results can be easily performed.

13.2 Test Problem - Explanation of Scripts

The following is a detailed explanation of the MaPS scripts. In this explanation, it is assumed that the MAPSHOME is MaPS and that the simulation run is conducted on a test problem in the MaPS/runs/test directory. When MaPS scripts are initiated by entering maps at the prompt, the following appears on the screen:

```
----------------------------------------
 MAPS script as of 06/12/00
----------------------------------------
current directory=/MaPS/runs/test < ------
PROJECTS directory not here.
Create PROJECTS directory? (OK):
```

Pressing return at this point creates a PROJECTS subdirectory

```
----------------------------------------
 PROJECT name -->test<-- What now?
 ----------------------------------------
1) Geometry
2) Mesh
3) BCIC
4) Simulation
5) Change project from -->test
6) Delete project -->test
7) Quit (*)
Choose (7) :
```

The above choices are interrelated. The geometry of the problem must be completed before setting the boundary conditions (BCIC) or meshing (Mesh) the problem. Simulation, which starts the conditions for the run, may be selected to either submit a run or examine results. Options 1-4 are discussed in detail below. Option 5 may be selected if the project must be changed, and Option 6 deletes the project *test* from the PROJECTS subdirectory. Option 7 quits MaPS scripts. In this and the subsequent scripts, an (*) beside an option indicates that it is the default for carriage return.

Choosing Option 1 gives

```
GEOMETRY SELECTED
GEOM.NE does NOT exist for  --- >test< ----
----------------------------------------
 GEOMETRY -->test<-- What now?
 ----------------------------------------
1) Patran
2) Geometry assignment
3) Set Defaults
```

4) Leave GEOMETRY (*)
Choose :

Choosing Option 2 assigns neutral file to be used for *test* geometry.

```
---------------------------------------
GEOM.NE already exists for -->test<--.
---------------------------------------
Assign neutral file to be used for test geometry.
Enter Neutral Filename (?) :
1) test1.NE
2) test2.NE
3) test3.NE
Choose number :

----------------------------
| Identify NEUTRAL file UNITS |
 ----------------------------
1) meters
2) centimeters
3) millimeters (*)
4) inches
5) feet
6) yards
Choose :
```

If the user wants to use the neutral file for the simulation, MaPS inquires about the units used while building the neutral file. Non-SI units used while generating a session file and, subsequently, a neutral file, are converted to SI units. Choosing Option 2 submits the test-HPVOL job, which calculates the hyperpatch volumes.

```
NEUTRAL file UNITS=centimeters (OK) :
UNITS file created
test.NE linked to GEOM.NE

Jobname                 : test-HPVOL
Load module             : /MaPS/bin/Hpvol
Neutral file in         : /MaPS/runs/test/PROJECTS/test/GEOM.NE
Hyperpatch volume file  : /MaPS/runs/test/PROJECTS/test/HPVOL
Standard output         : /MaPS/runs/test/test-HPVOL

(OK) to submit? :
```

Negating at this point gives the option of changing the defaults for the HPVOL run or leaving the HPVOL job together. On consent, the test-HPVOL job is submitted in background and the following is initiated:

```
HPVOL JOB MPs-11-HPVOL IN BACKGROUND id=18529
```

```
Erase all previous MESH, BCIC & MODEL info for -->test ? (OK) :
HPVOL JOB test-HPVOL COMPLETED id=18529

---------------------------------------
 GEOMETRY -->test<-- What now?
---------------------------------------
1) Patran
2) Geometry assignment
3) Set Defaults
4) Leave GEOMETRY (*)
Choose :
```

As mentioned earlier, the geometric assignment in MaPS can be initiated either immediately after saving the neutral file and consenting MaPS to use the neutral file for GEOM.NE or by choosing the Geometric Assignment option. At this point, the defaults of the load modules can be set by choosing the Set Defaults option and a consent at the prompt:

```
---------------------------------------
DEFAULTS
---------------------------------------
ODIR=/MaPS/runs/test
CLOAD=/MaPS/bin/Cas
CPLOAD=/MaPS/bin/Capspat
MLOAD=/MaPS/bin/Mesher
BCLOAD=/MaPS/bin/Bcgen
VMLOAD=/MaPS/bin/Vmesher
HPVLOAD=/MaPS/bin/HPvol
VBLOAD=/MaPS/bin/Vbounds
PSLOAD=/MaPS/bin/Print_step
MATLIB=/MaPS/bin/Properties
Change defaults? (OK) :
```

Here, ODIR is the output directory, CLOAD is the load module of MaPS, CPLOAD is the load module that links MaPS to PATRAN, MLOAD meshes the geometry by using either a name list data file or VMLOAD, BCLOAD generates the boundary conditions for the problem, VMLOAD is a visual interface that uses HOOPS libraries to visually mesh the geometry, HPVLOAD sets up the geometric linkage, VBLOAD is a visual interface that uses HOOPS libraries to visually set the boundary conditions and identify the named components, PSLOAD furnishes a printed output of the results of the simulation, and MATLIB contains the material-property libraries.

If the defaults are set correctly, the user answers with a "no" and obtains the GEOMETRY menu; otherwise, hitting return allows the user to change the defaults. The defaults can be set during geometry setting, meshing, or setting of boundary conditions. Also, as explained earlier, before submitting the job in background, the user has the option of either setting the defaults at that time or forfeiting the run. Option 4 allows the user to leave the geometry and get the root menu:

```
---------------------------------------
```

```
 PROJECT name -->test<-- What now?
 ----------------------------------------
1) Geometry
2) Mesh
3) BCIC
4) Simulation
5) Change project from -->test
6) Delete project -->test
7) Quit (*)
Choose (7) :
```

Now, with the geometry set, the user has the option of either meshing the geometry or setting up the boundary conditions. Choosing Option 3, which facilitates in setting the boundary conditions, gives

```
BCIC SELECTED
----------------------------------------
NO previous BCIC's found for -->test
Add new bcic -->BCIC.1<-- (OK) :
```

If this is the first run, the boundary conditions can be set under BCIC.1. The second time through, the user has the option of using BCIC.1 again or selecting BCIC.2 by negating at the above prompt. Thus, alternative boundary and initial conditions may be specified differently in BCIC.1, BCIC.2, etc., for each project. Consenting at the above prompt gives

```
INPUT does NOT exist for BCIC.1
 ----------------------------------------
 BCIC -->test<--
    -->BCIC.1<-- What now?
 ----------------------------------------
1) Visual BCIC
2) Assign input
3) Change BCIC from BCIC.1
4) Delete BCIC.1
5) Set Defaults
6) Leave BCIC (*)
Choose :
```

Option 1 instigates the VBLOAD module, which interacts with the HOOPS libraries and visually helps the user to set up the boundary conditions and also assigns the input. Option 2 allows the user to assign the input and is required if Option 1 is not selected. If more than one BCIC exists, the user can change the current BCIC by choosing Option 3. Option 4 deletes the currently activated BCIC, and Option 5 sets the defaults, as mentioned earlier. Option 6 takes the user back to the root menu. Choosing Option 1, the visual BCIC, gives

```
Visual BCIC selected.
Optionally enter BC input database (none) [?] :
```

If the BC dataset exists in the test directory, the user has the option of using it as input into the visual BCIC module. A "?" gives a listing of all the BC datasets in the test directory:

```
Visual BCIC load module          :/MaPS/bin/Vbounds
HPVOL in                         :/MaPS/runs/test/PROJECTS/test/HPVOL
Neutral file in                  :/MaPS/runs/test/PROJECTS/test/GEOM.ME
Output                           :/MaPS/runs/test/test-BCIC.1
BCIC directory                   :/MaPS/runs/test/PROJECTS/test/BCIC.1
AVAILABLE_MATERIALS              :/MaPS/properties/available_materials
```

Pressing return assumes that no BC data set is to be entered in the VBLOAD module and the visual boundary conditions setup screen is activated. Please refer to the related section on Vbounds in this manual. Creating and saving the input in Vbounds gives

```
A new BCIC INPUT file has been created
Save BCIC INPUT in current directory?  (OK)  :
```

A consent at this prompt gives

```
BCIC INPUT file will be saved as  -- >test.BC.1< --  (OK)  :
```

Note that if test.BC.1 exists, MaPS asks the user to choose a file to save the input or enter "n" so that the input file can be saved as test.BC.2, and so on:

```
Create  -- >test.BC.1< --
-- >test.BC.1< --  created.
INPUT exists for BCIC.1
 ---------------------------------------
  BCIC -->MPs-11<--
     -->BCIC.1<-- What now?
 ---------------------------------------
1) Visual BCIC
2) Assign input
3) Change BCIC from BCIC.1
4) Delete BCIC.1
5) Set Defaults
6) Leave BCIC (*)
Choose :
```

Another method of assigning the input data file to BCIC.1 is to choose Option 2. Finally, after the boundary and initial conditions are set, Option 6 can be chosen to obtain the root menu

```
 ---------------------------------------
  PROJECT name -->test<-- What now?
 ---------------------------------------
1) Geometry
```

2) Mesh
3) BCIC
4) Simulation
5) Change project from -->test
6) Delete project -->test
7) Quit (*)
Choose (7) :

Now, with the geometry and boundary conditions set, the user has the option of meshing the geometry. Choosing Option 2 facilitates setting the mesh for the geometry.

MESH SELECTED

NO previous MESHes found for -- >test
Add new mesh -- >MESH.1< -- (OK) :

If this is the first run, the new mesh can be set under MESH.1. The second time through, the user has the option of using MESH.1 again or of selecting MESH.2 by negating at the above prompt. In MaPS, a particular project can have several meshing alternatives. Alternate meshes for the same geometry are named sequentially as MESH.1, MESH.2, etc. Consenting at the above prompt gives

CELLDATA does not exist for -- >test< --
-- >MESH.1< --
MaPS does not exist for -- >test< --
-- >MESH.1< --
CELL.NE does not exist for -- >test< --
-- >MESH.1< --

MESH -->test<--
-->MESH.1<-- What now?
---
1) Visual mesher
2) Namelist mesher
3) PATRAN view mesh (NA)
4) Change MESH from MESH.1
5) Delete MESH.1
6) Set Defaults
7) Leave MESH (*)
Choose :

Option 1 initiates the visual mesher that interacts with the HOOPS libraries. Separate CAPSGEOMETYRY or MOLDGEOMETRY characteristics of the mesh can be viewed. Generation of the mesh creates the mesh input according to the conditions set in the visual mesher. Please refer to Section 10.3 on the Visual mesher in this manual for further explanation. Option 2 allows the user to use a name list file created by the Unix editor as input to the meshing module. Option 3 is not available in MaPS. If more than one MESH exists, the user can change the current

BCIC by selecting Option 4. Option 5 deletes the currently activated MESH.n, and Option 6 helps in changing the defaults for the mesh job. Option 7 returns the user to the root menu. Choosing Option 1, the visual mesher, gives

```
Visual mesher selected
Optionally enter ME input dataset (none) [?] :
```

If the ME dataset exists in the test directory, the user has the option of using that as input into the visual BCIC module. A "?" gives a listing of all BC datasets in the test directory. Pressing return assumes that no ME dataset is to be input in the Vmesher module, and the visual meshing setup screen is activated.

```
Visual mesher load module: /MaPS/bin/Vmesher
Mesher load module      : /MaPS/bin/Mesher
HPVOL in                : /MaPS/runs/test/PROJECTS/test/HPVOL
Neutral file in         : /MaPS/runs/test/PROJECTS/test/GEOM.NE
Neutral file out        : /MaPS/runs/test/PROJECTS/test/MESH.1/CELL.NE
Cell data out           : /MaPS/runs/test/PROJECTS/test/MESH.1/CELLDATA
Map data out            : /MaPS/runs/test/PROJECTS/test/MESH.1/MAP
Output                  : /MaPS/runs/test/test-MESH.1
BCIC directory          : /MaPS/runs/test/PROJECTS/test/MESH.1
```

Please refer to Section 10.3 on the Vmesher in this manual. Creating and saving the input in Vmesher gives

```
Save mesher INPUT? (OK) :
```

A consent at the prompt saves the input parameters entered in the visual meshing interface. Note that if test.ME.1 exists, MaPS asks the user to choose a file, to save the input, or enter "n" so the input file can be saved as test.ME.2, and so on.

```
Create -- >test.ME.1< --
-- >test.ME.1< -- created.

CELLDATA exists for     -- >test< --
                        -- >MESH.1< --
MaPS exists for         -- >test< --
                        -- >MESH.1< --
CELL.NE exists for      -- >test< --
                        -- >MESH.1< --
----------------------------------------
 MESH -->test<--
   -->MESH.1<-- What now?
----------------------------------------
1) Visual mesher
2) Namelist mesher
3) PATRAN view mesh
```

4) Change MESH from MESH.1
5) Delete MESH.1
6) Set Defaults
7) Leave MESH (*)
Choose :

Note that CELLDATA, MAP, and CELL.NE must all exist, indicating that mesh generation was successful. Finally, after the geometry is meshed, Option 7 can be chosen to obtain the root menu

---
PROJECT name -->test<-- What now?
---
1) Geometry
2) Mesh
3) BCIC
4) Simulation
5) Change project from -->test
6) Delete project -->test
7) Quit (*)
Choose (7) :

Now, because the geometry is built, the initial- and boundary-condition input parameters are set, and the geometry is meshed, the user is prepared to simulate the phenomena.

SIMULATION SELECTED

NO previous MODELs found for -->test
Add new model -->MODEL.1<-- (OK) :

If models exist, the user is allowed to choose the model number or a new model that involves a BCIC.n and a MESH.n. If no models exist, a consent at the above prompt gives the choice of the various MESH.n and BCIC.n that may have been created. If only one BCIC.1 and one MESH.1 exist, we obtain

---
Choose mesh number to use for MODEL.1 :
There is only one to choose -->MESH.1<-- (OK) :

Consent at the above prompt verifies if CELLDATA, MAP, and CELL.NE exist for MESH.1. If they exist, the user is then prompted to select BCIC.n:

CELLDATA exists for -->test<--
-->MESH.1<--
MAP exists for -->test<--
-->MESH.1<--
CELL.NE exists for -->test<--
-->MESH.1<--
MODEL.1 is using MESH.1

Choose BCIC number to use for MODEL.1 :
There is only one to choose -->BCIC.1<-- (OK) :

This prompt is activated only if one BCIC.n exists; otherwise, a list of all the BCICs is given for the user to choose from. A return at the above prompt verifies whether input exists for the selected BCIC and initiates the creation of /RUN.0/LASTSTEP

```
INPUT exists for -->test<--
          -->BCIC.1<--
MODEL.1 is using BCIC.1
MODEL.1/RUN.0/LASTSTEP does NOT exist.

Jobname              : test-MODEL.1-RUN.0
Load module          : /MaPS/ /bin/Bcgen
RUN data out         : /MaPS/runs/test/PROJECTS/test/MODEL.1/RUN.0
Neutral file in      : /MaPS/runs/test/PROJECTS/test/GEOM.NE
BC input data        : /MaPS/runs/test/PROJECTS/test/BCIC.1/INPUT
Cell data in         : /MaPS/runs/test/PROJECTS/test/MESH.1/CELLDATA
Map data in          : /MaPS/runs/test/PROJECTS/test/MESH.1/MAP
Materials Library    : /MaPS/materials_library
Standard output      : /MaPS/runs/test/test-MODEL.1-RUN.0

(OK) to submit NOW? [defer] :
```

Negating at the prompt allows the user to either change the defaults or leave the run. Consent at the prompt initiates the module that generates boundary conditions, Bcgen, and submits the RUN.0 job in background.

```
----------------------------------------
 SIMULATION      -->test<--
                 -->MODEL.1<--
                 -->BCIC.1<--
                 -->MESH.1<-- What now?
----------------------------------------
1) Run
2) Examine Results
3) Assign EMFDATA
4) Change MODEL from MODEL.1
5) Delete MODEL.1
6) Leave SIMULATION (*)
Choose :
```

Before going further, it is important that the Bcgen job that was submitted in the background be successfully completed. This is indicated by a completion prompt. The above choices can be selected, depending on whether a simulation run is to be conducted (Option 1) or if simulation results are to be examined (Option 2). The data (filename.EMF) of electrical resistivity heat used in Electroconsolidation is input by choosing Option 3. The model can be changed by selecting Option 4, and the current active model can be deleted by choosing Option 5. Option 6 again takes us to the

root menu. RUN.0 and future runs can be examined via Option 2, which helps in analyzing the simulation results. Choosing Option 3 gives

```
---------------------------------------
Assign EMFDATA selected.
MODEL.1/RUN.0/EMFDATA does NOT exist.
---------------------------------------
Enter EMF dataset (?) :
---------------------------------------
```

A "?" gives a listing of all the *EMF data sets in the test directory. If more than one input dataset exists, the user is asked to choose one of them; otherwise only one is listed upon entering "?." This is a required step in the Electroconsolidation process. After assigning the *EMF data set

test.EMF linked to MODEL.1/RUN.0/EMFDATA

```
---------------------------------------
 SIMULATION -->test<--
        -->MODEL.1<--
        -->BCIC.1<--
        -->MESH.1<-- What now?
 ---------------------------------------
1) Run
2) Examine Results
3) Assign EMFDATA
4) Change MODEL from MODEL.1
5) Delete MODEL.1
6) Leave SIMULATION (*)
Choose :
```

Choosing Option 1 gives

```
RUN selected.
NO previous RUN's found for -->test-MODEL.1
Add new run -->RUN.1<-- (OK) :
```

If runs other than RUN.0 exist, the user is queried about which run to choose. Assuming that no other runs exist, a consent at the above prompt gives

```
---------------------------------
 RUN     -->test<--
         -->MODEL.1<--
         -->MESH.1<--
         -->BCIC.1<--
         -->RUN.1<-- What now?
 ---------------------------------
1) Submit run
2) Change RUN from RUN.1
3) Delete RUN.1
```

4) Leave RUN (*)
Choose :

Option 1 initiates the conditions to be set to submit a simulation run. If more than one run exists, Option 2 can be chosen to change the run. Option 3 deletes the run. Choosing Option 1 gives

SUBMIT RUN SELECTED.
Enter input TS dataset (?) :

Here, the *TS filename is entered to initiate RUN.1. If more than one input dataset exists, the user is asked to choose one of them; otherwise only one is listed up on entering "?." Once the input *TS file is selected, the user is queried if any other run is to be continued. The user can restart a previous run by choosing a directory of that run to be entered in the present run. If no previous runs exist for the present project, only RUN.0 is given as a choice for input. Note that RUN.0 is the initial state and could be used as an input to all runs.

Choose directory to use for input or "RETURN" for RUN.0 :
There is only one to choose -->RUN.0<-- (OK) :

Consent at the above prompt gives the option of choosing the step number at which a previous run must be restarted. RUN.0, if selected, only has step.00000 available as input.

Enter step number (000000) [?] :

Jobname	: test-MODEL.1-RUN.1
Load module	: /MaPS/bin/Cas
RUN data out	: /MaPS/runs/test /PROJECTS/test/MODEL.1/RUN.1
TS input data	: /MaPS/runs/test/test.TS.1
Cell data in	: /MaPS/runs/test /PROJECTS/test/MESH.1/CELLDATA
RUN data in	: /MaPS/runs/test /PROJECTS/test/MODEL.1/RUN.0
Step used	: 000000
Standard output	: /MaPS/runs/test /test-MODEL.1-RUN.1

(OK) to submit NOW? [defer] :

Upon entering the step number or choosing the default step.000000 from RUN.0, RUN.1 is submitted in background upon consenting at the prompt; otherwise the user has the choice of leaving the run or changing the defaults, as previously mentioned. The following scripts are then activated:

test-RUN.1 COMPLETED

Leave SIMULATION selected.

```
PROJECT name -->test<-- What now?
---------------------------------------
1) Geometry
2) Mesh
3) BCIC
4) Simulation
5) Change project from -->MPs-30
6) Delete project -->MPs-30
7) Quit (*)
Choose (7) : 4
```

Choosing Option 4 will initiate another simulation run.

```
SIMULATION SELECTED
---------------------------------------
Use MODEL.1 ? (OK) :
---------------------------------------
MODEL.1 is using MESH.1
MODEL.1 is using BCIC.1
-----------------------------------------------
 SIMULATION        -->test<--
                   -->MODEL.1<--
                   -->MESH.1<--
                   -->BCIC.1<--
-----------------------------------------------
1) Run
2) Examine Results
3) Assign EMFDATA
4) Change MODEL from MODEL.1
5) Delete MODEL.1
6) Leave SIMULATION (*)
Choose : 3
```

It must be emphasized that EMFDATA must be entered again to restart of the previous run. Without doing this input in the restart job of Electroconsolidation, the electrical resistivity heat will be assumed to be zero in the system. Option 3 issues the script as

```
-----------------------------------------------------
Assign EMFDATA selected.
MODEL.1/RUN.0/EMFDATA already EXISTS
-----------------------------------------------------
Delete it? (No): yes
```

We reminded the user that EMFDATA exists only in the RUN.0 directory, not in the RUN.1 directory. Therefore, the user must enter "yes" to have the opportunity to input the EMFDATA in the RUN.1 directory. The "yes" command gives the script

```
-------------------------------------------------------------------
MODEL.1/RUN.0/EMFDATA     <----- DELETED!
```

```
------------------------------------------------------
Enter EMF dataset (?) : ?
------------------------------------------------------
Choose EMF dataset :
------------------------------------------------------
1) test.1.EMF
2) test.2.EMF
3) test.3.EMF
Choose number :
```

After choosing the number of EMF data sets required for the restart job, the screen shows

```
------------------------------------------------
SIMULATION      -->test<--
                -->MODEL.1<--
                -->MESH.1<--
                -->BCIC.1<--
------------------------------------------------
1) Run
2) Examine Results
3) Assign EMFDATA
4) Change MODEL from MODEL.1
5) Delete MODEL.1
6) Leave SIMULATION (*)
Choose : 1
```

Option 1 starts a new run or continues another run as a restart from the previous run. Option 2 allows the user to use another number for the next run. Option 3 allows the user to delete an existing run directory, and Option 4 exits the run simulation.

```
RUN selected.
Last was  -->RUN.1<--  Reuse it ? No
Choose run number or Enter "n" for next :
There is only one to choose  -->RUN.1<--  (OK) : No
Add new run  -->RUN.2<--  (OK)  : OK

------------------------------------------------
RUN             -->test<--
                -->MODEL.1<--
                -->MESH.1<--
                -->BCIC.1<--
                -->RUN.2<--   What now?
------------------------------------------------
1) Submit run
2) Change RUN from RUN.2
3) Delete RUN.2
4) Leave RUN (*)
Choose : 1

SUBMIT RUN SELECTED.
Enter input TS dataset (?) :
```

```
Choose input TS dataset :
1) test.TS.1
2) test.TS.2
Choose number : 2
```

Here, the *TS filename is entered to initiate a new RUN.2 or a continued RUN.2 as a restart from the previous RUN.1. If more than one input dataset exists, the user is asked to choose one of them. To continue the previous run, the user chooses one that is correct to restart the job. Once the input *TS file is selected, the user is queried if any other run is to be continued. The user can restart a previous run by choosing a directory of that run (e.g., RUN.1) to be input to the present run. If no previous runs exist for the present project, only RUN.0 is given as a choice for input. Note that RUN.0 is the initial state and could be used as an input to all runs.

```
Enter jobname (test-MODEL.1-RUN.2) :
Choose directory to use for input or "RETURN" for RUN.0 :
1) RUN.0
2) RUN.1
Choose number : 2
Enter step number (time step *** of previous run) [?] :

Jobname          : test-MODEL.1-RUN.2
Load module      : /MaPS /bin/Cas
RUN data out     : /MaPS/runs/test/PROJECTS/test/MODEL.1/RUN.2
TS input data    : test.TS.2
Cell data in     : /MaPS/runs/test/PROJECTS/test MESH.1/CELLDATA
RUN data in      : /MaPS/runs/test/PROJECTS/test/MODEL.1/RUN.1
Step used        : (time step of previous run)
Standard output  : /MaPS/runs/test/test-MODEL.1-RUN.2

(OK) to submit NOW? [defer] :
```

A consent at the above prompt activates the run job test-MODEL.1-RUN.2 starting in background; otherwise, the user has the choice of leaving the run or changing the defaults, as previously mentioned. The following scripts are then activated:

```
-------------------------------------------------
 SIMULATION      -->test<--
                 -->MODEL.1<--
                 -->MESH.1<--
                 -->BCIC.1<--
-------------------------------------------------
1) Run
2) Examine Results
3) Assign EMFDATA
4) Change MODEL from MODEL.1
5) Delete MODEL.1
6) Leave SIMULATION (*)
Choose :
```

Choosing Option 6 will exit MaPS.

14 Summary

A coupled-field computer model that utilizes ELEKTRA and MaPS has been developed for the analysis of temperature distribution in Electroconsolidation performed by the powder industry. Insight into the Electroconsolidation process via computer modeling will provide a better understanding of the Electroconsolidation process. Also, initial analysis will save dollars in terms of energy and time usage. Thus, by using computer modeling, the process can be simplified before the settings are made and before consolidating powder parts.

In addition to being the basis for a real-time feedback control system, the computer model can be used to estimate power and setup requirements in Electroconsolidation. The model can also be used to plan the placement of multiple preforms in a die to minimize potential hot spots during consolidation and therefore to optimize the design of the Electroconsolidation process.

The computer model has been tested and verified in parallel with the development. Extensive simulations have been conducted to check and verify every step of the modeling. Thus, it is generally felt that this developed model is a very well tested simulation tool. We certainly welcome feedback from any user who may encounter a bug or who has suggestions for improvements of any kind.

Acknowledgments

This work was sponsored by the U.S. Department of Commerce, National Institute of Standards and Technology, Advanced Technology Program, and by Superior Graphite Company. The authors are grateful for the efforts of Dr. Raymond Fessler of Biztek Consulting. Sincere gratitude and thanks are expressed to Dr. William Goldberger and Senior Research Engineers Brian Merkle and Josh Borton, all of Superior Graphite, for their guidance and support in developing and validating the computer model.

References

1. W. M. Goldberger, B. D. Merkle, and D. Boss, Making Dense Near Net Shaped Parts, Advanced Processing Techniques – Particulate Materials, 1994, Vol. 6, pp. 91-102.

2. W. M. Goldberger, A Low Cost Method for Pressure-Assisted Densification of Advanced Materials into Complex Shaped Parts, Materials Technology, 1995, Vol. 10 (3/4), pp. 48-51.

3. R. M. German, A Review of Numerical and Computational Techniques in Powder Metallurgy, Advances in Powder Metallurgy & Particulate Materials, 1993, Vol. 3, pp. 1-18.

4. B. L. Ferguson and A. Lawley, Modeling of Powder Metallurgy Processes, *Advances in* Powder Metallurgy & Particulate Materials, 1993, Vol. 3, pp. 19-44.

5. T. F. Zahrah and L. Christodoulou, Modeling and Design of P/M consolidation Processes, Advances in Powder Metallurgy & Particulate Materials, 1993, Vol. 3, pp. 77-83.

6. J. R. L. Trasorras, S. Armstrong, and T. J. McCabe, Modeling the Compaction of Steel Powder Parts, Advances in Powder Metallurgy & Particulate Materials, 1994, Vol. 7, pp. 33-50.

7. H. M. Domanus, R. C. Schmitt, and S. Ahuja, User's Guide for the Casting Process Simulator Software CaPS-2D, Version 1.0, Argonne National Laboratory Report ANL-93/14, July 1993.

8. F. C. Chang, Modeling of MHD Edge containment in Strip Casting with ELEKTRA and CaPS-EM Codes, Argonne National Laboratory Report ANL-00/1, Feb. 2000.

9. OPERA-3D Reference Manual - Software for Electromagnetic Design, published by Vector Fields Ltd., Oxford, England, 1998.

10. OPERA-3D User Guide - Software for Electromagnetic Design, published by Vector Fields Ltd., Oxford, England, 1998.

Appendix A: File Structure Overview

The following is a listing of all input and output files created by the user of MaPS. The files are, therefore, linked by the MaPS shell scripts.

A.1 Current Working Directory

PROJECT/	Directory used to organize the data files used in performing MaPS simulations.
*.NE.n	Optionally saved CPRE neutral files.
*.pre.n	Preprocess file for geometry and named components of MaPS.
*.ME.n	Optionally saved files that contains free-form mesh control information used for meshing input.
*.BC.n	Optionally saved files that contain input for assigning boundary and initial conditions.
*.TS.n	Run control files.
*.comi	File automatically created by MaPS to read data (e.g., resistivity heat in Electroconsolidation) from ELEKTRA.
*.EMF	Input data of resistivity heat (read from ELEKTRA) to MaPS.
*.ps.n	POSTSCRIPT graphics files.
*.BUFFER.X/	Temporary directories used to postprocess MaPS results.
*-MODEL.n-RUN.n	File that contains the printed output from "Cas" execution.

A.2 PROJECTS Directory

.DEFAULTS	File that contains names of executable MaPS modules (hidden).
.PLAST	Files that contain name of last-used project (hidden).
Projectname/	Any number of specific project directories, each named by the user and referred to here generically as Projectname. Each project is distinguished by a unique geometry specification.

A.3 Projectname Directories

BCIC.n/	Boundary and initial condition assignment directories.
.DEFAULTS	File that contains names of executable MaPS modules (hidden).

GEOM.NE	Neutral file that contains geometric description of project.
HPVLOAD	Link to MaPS "Hpvol" executable module. Once a geometry has been defined and units have been declared, a job is submitted to preprocess the geometry and determine information to be used later by other MaPS modules.
HPVOL	File, generated by HPVLOAD, that contains information to be used by other load modules.
HPVOUTPUT	File that contains printed output from HPVLOAD execution.
LASTBCIC	File that contains name of most recently accessed initial and boundary conditions.
LASTMESH	Files that contains name of most recently accessed mesh.
LASTMODEL	File that contains name of last model that was accessed.
MESH.n/	Mesh directories.
MODEL.n/	Model directories. Each model may have a unique combination of meshing and boundary and initial condition assignments.
NEUTRALIN	Link to geometry neutral file (GEOM.NE) used as input to HPVLOAD.
UNITS	File that contains unit conversion name (inches, meters, etc.)

A.4 MESH.n Directories

CELL.NE	Neutral file that contains mesh determined by MaPS mesher.
CELLDATA	File that contains binary cell and surface element data that define mesh that approximates project geometry.
HPVOL	Link to the file that contains preprocessed project geometry information.
INPUT	File that contains input mesh control information.
LASTMESH	Link to name of current mesh.
MAP	File that contains binary lists that relate cells and surface element to project geometry.
MLOAD	Link to MaPS Mesher-executable module.
MOUTPUT	File that contains printed output from mesher execution.
NEUTRALIN	Link to project geometry neutral file.

PLAST	Link to project name.
VMLOAD	Link to MaPS Vmesher-executable module.
VMOUTPUT	File that contains printed output from Vmesher execution.

A.5 BCIC.n Directories

AVAILABLE_MATERIALS	Link to file that contains a list of all property database files.
HPVOL	Link to file that contains preprocessed project geometry information.
INPUT	File that contains free-form boundary and initial condition assignments.
LASTBCIC	File that contains name of boundary and initial condition assignment associated with model.
NEUTRALIN	Link to project geometry neutral file.
PLAST	Link to project name.
VBLOAD	Link to MaPS Vbounds-executable module.
VBOUTPUT	File that contains printed output from Vbound execution.

A.6 MODEL.n Directories

LASTMESH	File that contains name of the mesh associated with model.
LASTBCIC	File that contains name of boundary and initial condition assignment associated with model.
LASTRUN	File that contains name of most recently accessed run.
RUN.0/	Directory that contains initial state of simulation, determined from mesh and boundary and initial condition assignments.
RUN.n/	Directories that contains raw results from MaPS simulation.

A.7 RUN.0 Directory

BCLOAD	Link to MaPS Bcgen-executable module.
CELLDATA	Link to file that contains binary cell and surface element data that define mesh that approximates project geometry.
EMFDATA	Data file that contains coordinates of cells to read data (e.g., resistivity heat in Electroconsolidation) from ELEKTRA.

INPUT	Link to file that contains free-form boundary and initial condition assignments.
INVAROUT	Binary file that contains information that is invariant during a run.
LASTSTEP	File that contains the number "000000," indicating last step in RUN.0 directory.
MAP	Link to file that contains binary lists that relate cells and surface elements to project geometry.
MATLIB	Link to directory that contains properties library.
NEUTRALIN	Link to project geometry neutral file.
OUTPUT	File that contains printed output from Bcgen execution.
Step.000000	Binary file that contains variable array values.
STEPLIST	File that contains name "step.000000," present in RUN.0 directory.

A.8 RUN.n Directories

CLOAD	Link to MaPS Cas-executable module.
CELLDATA	Link to file that contains binary cell and surface element data that define mesh that approximates project geometry.
EMFDATA	Data file that contains coordinates of cells to read data (e.g., resistivity heat in Electroconsolidation) from ELEKTRA.
INPUT	Link to run control file.
INVARIN	Link to binary file that contains information that was invariant during run used to initialize current run.
INVAROUT	Binary file that contains information that is invariant during current run.
LASTSTEP	File that contains number of last step in current RUN.n directory.
MONITOROUT	Currently unused.
OUTPUT	File that contains printed output from Cas execution.
SELECT.dat	Data file that contains printed output information (e.g., temperature in Electroconsolidation) selected in *.TS.n file.

STEPIN	Link to file that contains variable array values chosen from run used to initialize current run.
STEP.nnnnnn	Binary files that contain variable array values for timestep number "nnnnnn."
STEPLIST	File that contains list of the names "step.nnnnnn" in current run directory.

Appendix B: Samples of MaPS Preprocess File

Sample 1:

```
ibug=0
c-------------------------------------------------------------------------
c preprocessor input for Superior Graphite Electroconsolidation
c Single preform, only electrical insulation around the die
c Only movement at upper ram
c Unit: cm
c-------------------------------------------------------------------------

grid  1=(    0.0,     0.0, -3.302)
grid  2=(   3.81,     0.0, -3.302)
grid  3=(   3.52,   1.458, -3.302)
grid  4=(  2.694,   2.694, -3.302)
grid  5=(  1.458,    3.52, -3.302)
grid  6=(    0.0,    3.81, -3.302)

grid  7=(    0.0,     0.0,  3.302)
grid  8=(   3.81,     0.0,  3.302)
grid  9=(   3.52,   1.458,  3.302)
grid 10=(  2.694,   2.694,  3.302)
grid 11=(  1.458,    3.52,  3.302)
grid 12=(    0.0,    3.81,  3.302)

hyperpatch 1=( 1, 3, 2, 1, 7,  9,  8, 7)   medium
hyperpatch 2=( 1, 4, 3, 1, 7, 10,  9, 7)
hyperpatch 3=( 1, 5, 4, 1, 7, 11, 10, 7)
hyperpatch 4=( 1, 6, 5, 1, 7, 12, 11, 7)

c Grid for Ram

grid 13=(    0.0,     0.0, -16.002)
grid 14=(   3.81,     0.0, -16.002)
grid 15=(   3.52,   1.458, -16.002)
grid 16=(  2.694,   2.694, -16.002)
grid 17=(  1.458,    3.52, -16.002)
grid 18=(    0.0,    3.81, -16.002)

grid 19=(    0.0,     0.0,  16.002)
grid 20=(   3.81,     0.0,  16.002)
grid 21=(   3.52,   1.458,  16.002)
grid 22=(  2.694,   2.694,  16.002)
grid 23=(  1.458,    3.52,  16.002)
grid 24=(    0.0,    3.81,  16.002)

hyperpatch 5=( 13, 15, 14, 13,  1,  3,  2,  1)   Lower Ram
hyperpatch 6=( 13, 16, 15, 13,  1,  4,  3,  1)
hyperpatch 7=( 13, 17, 16, 13,  1,  5,  4,  1)
hyperpatch 8=( 13, 18, 17, 13,  1,  6,  5,  1)

hyperpatch 9 =( 7,  9,  8,  7, 19, 21, 20, 19)   Upper Ram
```

```
hyperpatch 10=( 7, 10,  9,  7, 19, 22, 21, 19)
hyperpatch 11=( 7, 11, 10,  7, 19, 23, 22, 19)
hyperpatch 12=( 7, 12, 11,  7, 19, 24, 23, 19)

c Grid for Die Chamber

grid 25=(    3.81,     0.0, -9.2705)
grid 26=(    3.52,   1.458, -9.2705)
grid 27=(   2.694,   2.694, -9.2705)
grid 28=(   1.458,    3.52, -9.2705)
grid 29=(     0.0,    3.81, -9.2705)

grid 30=(    8.89,     0.0, -9.2705)
grid 31=(   8.213,   3.402, -9.2705)
grid 32=(   6.286,   6.286, -9.2705)
grid 33=(   3.402,   8.213, -9.2705)
grid 34=(     0.0,    8.89, -9.2705)

grid 35=(    3.81,     0.0,  8.5725)
grid 36=(    3.52,   1.458,  8.5725)
grid 37=(   2.694,   2.694,  8.5725)
grid 38=(   1.458,    3.52,  8.5725)
grid 39=(     0.0,    3.81,  8.5725)

grid 40=(    8.89,     0.0,  8.5725)
grid 41=(   8.213,   3.402,  8.5725)
grid 42=(   6.286,   6.286,  8.5725)
grid 43=(   3.402,   8.213,  8.5725)
grid 44=(     0.0,    8.89,  8.5725)

hyperpatch 13=( 25, 26, 31, 30, 35, 36, 41, 40)   Die Chamber
hyperpatch 14=( 26, 27, 32, 31, 36, 37, 42, 41)
hyperpatch 15=( 27, 28, 33, 32, 37, 38, 43, 42)
hyperpatch 16=( 28, 29, 34, 33, 38, 39, 44, 43)

c grids for patch and mesh control

grid 45=(     0.0,     0.0, -9.2075)
grid 46=(     0.0,     0.0,   -0.05)
grid 47=(     0.0,     0.0,  -0.025)
grid 48=(     0.0,     0.0,     0.0)
grid 49=(     0.0,     0.0,   0.025)
grid 50=(     0.0,     0.0,    0.05)
grid 51=(     0.0,     0.0,  8.5725)

name=CAPSGEOMETRY hp=( 1, 2, 3, 4, 5, 6, 7, 8, 9,10,
                       11,12,13,14,15,16)

name=MEDIUM       hp=( 1, 2, 3, 4)

name=RAM          hp=( 5, 6, 7, 8, 9,10,11,12)

name=DIE          hp=(13,14,15,16)
```

```
patch  1=( 13, 15, 14, 13)        Lower Ram Bottom Surface
patch  2=( 13, 16, 15, 13)
patch  3=( 13, 17, 16, 13)
patch  4=( 13, 18, 17, 13)

patch  5=( 19, 21, 20, 19)        Upper Ram Top Surface
patch  6=( 19, 22, 21, 19)
patch  7=( 19, 23, 22, 19)
patch  8=( 19, 24, 23, 19)

patch  9 =( 14, 15, 26, 25)       Lower Ram Side Surface
patch 10=( 15, 16, 27, 26)
patch 11=( 16, 17, 28, 27)
patch 12=( 17, 18, 29, 28)

patch 13=( 35, 36, 21, 20)        Upper Ram Side Surface
patch 14=( 36, 37, 22, 21)
patch 15=( 37, 38, 23, 22)
patch 16=( 38, 39, 24, 23)

patch 17=( 25, 26, 31, 30)        Die Bottom Surface
patch 18=( 26, 27, 32, 31)
patch 19=( 27, 28, 33, 32)
patch 20=( 28, 29, 34, 33)

patch 21=( 35, 36, 41, 40)        Die Top Surface
patch 22=( 36, 37, 42, 41)
patch 23=( 37, 38, 43, 42)
patch 24=( 38, 39, 44, 43)

patch 25=( 30, 31, 41, 40)        Die Side Surface
patch 26=( 31, 32, 42, 41)
patch 27=( 32, 33, 43, 42)
patch 28=( 33, 34, 44, 43)

patch 29=( 13, 14, 20, 19)        Central Surface
patch 30=( 13, 18, 24, 19)
patch 31=( 25, 30, 40, 35)
patch 32=( 29, 34, 44, 39)

name=LRAMBS    patch=( 1, 2, 3, 4)

name=URAMTS    patch=( 5, 6, 7, 8)

name=RAMS      patch=( 9,10,11,12,13,14,15,16)

name=DIES      patch=(17,18,19,20,21,22,23,24,
                      25,26,27,28)

name=CENTER    patch=(29,30,31,32)

name=MESH_CONTROL grid=(  1,  7, 13, 19, 45, 46, 47,
                         48, 49, 50, 51,
                          2,  3,  4,  5,  6, 30, 34)
```

Sample 2:

```
ibug=0
c-------------------------------------------------------------------
c preprocessor input for Superior Graphite Electroconsolidation
c Multi preforms, only electrical insulation between filcarb
c and rams
c Unit: cm
c-------------------------------------------------------------------

c Grids for preform

grid  1=(      0.0,      0.0, -3.81)
grid  2=(  1.1255,      0.0, -3.81)
grid  3=(  1.1255,   1.1255, -3.81)
grid  4=(      0.0,   1.1255, -3.81)

grid  5=(      0.0,      0.0, -2.54)
grid  6=(  1.1255,      0.0, -2.54)
grid  7=(  1.1255,   1.1255, -2.54)
grid  8=(      0.0,   1.1255, -2.54)

grid  9=(      0.0,      0.0, -0.635)
grid 10=(  1.1255,      0.0, -0.635)
grid 11=(  1.1255,   1.1255, -0.635)
grid 12=(      0.0,   1.1255, -0.635)

grid 13=(      0.0,      0.0,  0.635)
grid 14=(  1.1255,      0.0,  0.635)
grid 15=(  1.1255,   1.1255,  0.635)
grid 16=(      0.0,   1.1255,  0.635)

grid 17=(      0.0,      0.0,  2.54)
grid 18=(  1.1255,      0.0,  2.54)
grid 19=(  1.1255,   1.1255,  2.54)
grid 20=(      0.0,   1.1255,  2.54)

grid 21=(      0.0,      0.0,  3.81)
grid 22=(  1.1255,      0.0,  3.81)
grid 23=(  1.1255,   1.1255,  3.81)
grid 24=(      0.0,   1.1255,  3.81)

hyperpatch 1=  1,  4,  3,  2,  5,  8,  7,  6)  Preforms
hyperpatch 2=(  9, 12, 11, 10, 13, 16, 15, 14)
hyperpatch 3=( 17, 20, 19, 18, 21, 24, 23, 22)

c Grids for powder-bt

grid 25=(      0.0,      0.0,  -5.715)
grid 26=(  1.1255,      0.0,  -5.715)
grid 27=(  1.1255,   1.1255,  -5.715)
grid 28=(      0.0,   1.1255,  -5.715)
```

```
grid 29=(      0.0,      0.0,  5.715)
grid 30=(  1.1255,      0.0,  5.715)
grid 31=(  1.1255,  1.1255,  5.715)
grid 32=(      0.0,  1.1255,  5.715)

hyperpatch  4=( 25, 28, 27, 26,  1,  4,  3,  2)  Powder-bt
hyperpatch  5=(  5,  8,  7,  6,  9, 12, 11, 10)
hyperpatch  6=( 13, 16, 15, 14, 17, 20, 19, 18)
hyperpatch  7=( 21, 24, 23, 22, 29, 32, 31, 30)

c Grids for powder-s

grid 33=(    6.35,      0.0,  -5.715)
grid 34=(   5.867,    2.430,  -5.715)
grid 35=(    4.49,     4.49,  -5.715)
grid 36=(   2.430,    5.867,  -5.715)
grid 37=(      0.0,    6.35,  -5.715)

grid 38=(    6.35,      0.0,   5.715)
grid 39=(   5.867,    2.430,   5.715)
grid 40=(    4.49,     4.49,   5.715)
grid 41=(   2.430,    5.867,   5.715)
grid 42=(      0.0,    6.35,   5.715)

hyperpatch  8=( 33, 26, 27, 34, 38, 30, 31, 39)  Powder-s
hyperpatch  9=( 34, 27, 36, 35, 39, 31, 41, 40)
hyperpatch 10=( 36, 27, 28, 37, 41, 31, 32, 42)

c Grids for bottom ram

grid 43=(      0.0,      0.0, -18.5737)
grid 44=(  1.1255,      0.0, -18.5737)
grid 45=(  1.1255,  1.1255, -18.5737)
grid 46=(      0.0,  1.1255, -18.5737)

grid 47=(    6.35,      0.0, -18.5737)
grid 48=(   5.867,    2.430, -18.5737)
grid 49=(    4.49,     4.49, -18.5737)
grid 50=(   2.430,    5.867, -18.5737)
grid 51=(      0.0,    6.35, -18.5737)

hyperpatch 11=( 43, 46, 45, 44, 25, 28, 27, 26)  Bottom ram
hyperpatch 12=( 47, 44, 45, 48, 33, 26, 27, 34)
hyperpatch 13=( 48, 45, 50, 49, 34, 27, 36, 35)
hyperpatch 14=( 50, 45, 46, 51, 36, 27, 28, 37)

c Grids for top ram

grid 52=(      0.0,      0.0,  18.5737)
grid 53=(  1.1255,      0.0,  18.5737)
grid 54=(  1.1255,  1.1255,  18.5737)
grid 55=(      0.0,  1.1255,  18.5737)

grid 56=(    6.35,      0.0,  18.5737)
```

```
grid 57=(   5.867,   2.430,  18.5737)
grid 58=(    4.49,    4.49,  18.5737)
grid 59=(   2.430,   5.867,  18.5737)
grid 60=(     0.0,    6.35,  18.5737)

hyperpatch 15=( 29, 32, 31, 30, 52, 55, 54, 53)  Top ram
hyperpatch 16=( 38, 30, 31, 39, 56, 53, 54, 57)
hyperpatch 17=( 39, 31, 41, 40, 57, 54, 59, 58)
hyperpatch 18=( 41, 31, 32, 42, 59, 54, 55, 60)

c grids for filcarb

grid 61=(    6.35,     0.0, -10.795)
grid 62=(   5.867,   2.430, -10.795)
grid 63=(    4.49,    4.49, -10.795)
grid 64=(   2.430,   5.867, -10.795)
grid 65=(     0.0,    6.35, -10.795)

grid 66=(   10.16,     0.0, -10.795)
grid 67=(   9.387,   3.888, -10.795)
grid 68=(   7.184,   7.184, -10.795)
grid 69=(   3.888,   9.387, -10.795)
grid 70=(     0.0,   10.16, -10.795)

grid 71=(    6.35,     0.0,  10.795)
grid 72=(   5.867,   2.430,  10.795)
grid 73=(    4.49,    4.49,  10.795)
grid 74=(   2.430,   5.867,  10.795)
grid 75=(     0.0,    6.35,  10.795)

grid 76=(   10.16,     0.0,  10.795)
grid 77=(   9.387,   3.888,  10.795)
grid 78=(   7.184,   7.184,  10.795)
grid 79=(   3.888,   9.387,  10.795)
grid 80=(     0.0,   10.16,  10.795)

hyperpatch 19=( 61, 62, 67, 66, 71, 72, 77, 76)  Filcarb
hyperpatch 20=( 62, 63, 68, 67, 72, 73, 78, 77)
hyperpatch 21=( 63, 64, 69, 68, 73, 74, 79, 78)
hyperpatch 22=( 64, 65, 70, 69, 74, 75, 80, 79)

c grids for felt

grid 81=(    6.35,     0.0,  -12.7)
grid 82=(   5.867,   2.430,  -12.7)
grid 83=(    4.49,    4.49,  -12.7)
grid 84=(   2.430,   5.867,  -12.7)
grid 85=(     0.0,    6.35,  -12.7)

grid 86=(   11.43,     0.0,  -12.7)
grid 87=(   10.56,   4.374,  -12.7)
grid 88=(   8.082,   8.082,  -12.7)
grid 89=(   4.374,   10.56,  -12.7)
grid 90=(     0.0,   11.43,  -12.7)
```

```
grid  91=(   11.43,     0.0, -10.795)
grid  92=(   10.56,   4.374, -10.795)
grid  93=(   8.082,   8.082, -10.795)
grid  94=(   4.374,   10.56, -10.795)
grid  95=(     0.0,   11.43, -10.795)

hyperpatch 23=( 81, 82, 87, 86, 61, 62, 92, 91)   Bottom felt
hyperpatch 24=( 82, 83, 88, 87, 62, 63, 93, 92)
hyperpatch 25=( 83, 84, 89, 88, 63, 64, 94, 93)
hyperpatch 26=( 84, 85, 90, 89, 64, 65, 95, 94)

grid  96=(    6.35,     0.0,    12.7)
grid  97=(   5.867,   2.430,    12.7)
grid  98=(    4.49,    4.49,    12.7)
grid  99=(   2.430,   5.867,    12.7)
grid 100=(     0.0,    6.35,    12.7)

grid 101=(   11.43,     0.0,    12.7)
grid 102=(   10.56,   4.374,    12.7)
grid 103=(   8.082,   8.082,    12.7)
grid 104=(   4.374,   10.56,    12.7)
grid 105=(     0.0,   11.43,    12.7)

grid 106=(   11.43,     0.0,  10.795)
grid 107=(   10.56,   4.374,  10.795)
grid 108=(   8.082,   8.082,  10.795)
grid 109=(   4.374,   10.56,  10.795)
grid 110=(     0.0,   11.43,  10.795)

hyperpatch 27=( 71, 72,107,106, 96,  97,102,101)   Top felt
hyperpatch 28=( 72, 73,108,107, 97,  98,103,102)
hyperpatch 29=( 73, 74,109,108, 98,  99,104,103)
hyperpatch 30=( 74, 75,110,109, 99,100,105,104)

hyperpatch 31=( 66, 67, 92, 91, 76, 77,107,106)   Side felt
hyperpatch 32=( 67, 68, 93, 92, 77, 78,108,107)
hyperpatch 33=( 68, 69, 94, 93, 78, 79,109,108)
hyperpatch 34=( 69, 70, 95, 94, 79, 80,110,109)

c grids for patch and mesh control

grid 111=(     0.0,     0.0,    -12.7)
grid 112=(     0.0,     0.0, -10.795)
grid 113=(     0.0,     0.0,    -5.08)
grid 114=(     0.0,     0.0,   -4.445)
grid 115=(     0.0,     0.0,    -3.39)
grid 116=(     0.0,     0.0,    -2.96)
grid 117=(     0.0,     0.0,   -1.905)
grid 118=(     0.0,     0.0,    -1.27)
grid 119=(     0.0,     0.0,   -0.215)
grid 120=(     0.0,     0.0,    0.215)
grid 121=(     0.0,     0.0,     1.27)
```

```
grid 122=(    0.0,    0.0,   1.905)
grid 123=(    0.0,    0.0,    2.96)
grid 124=(    0.0,    0.0,    3.39)
grid 125=(    0.0,    0.0,   4.445)
grid 126=(    0.0,    0.0,    5.08)
grid 127=(    0.0,    0.0,  10.795)
grid 128=(    0.0,    0.0,    12.7)

name=CAPSGEOMETRY hp=( 1, 2, 3, 4, 5, 6, 7, 8, 9,10,
                      11,12,13,14,15,16,17,18,19,20,
                       21,22,23,24,25,26,27,28,29,30,
                      31,32,33,34)

name=PREFORM      hp=( 1, 2, 3)

name=POWDER-BT    hp=( 4, 5, 6, 7)

name=POWDER-S     hp=( 8, 9,10)

name=RAM          hp=(11,12,13,14,15,16,17,18)

name=FILCARB      hp=(19,20,21,22)

name=FELT         hp=(23,24,25,26,27,28,29,30,
                      31,32,33,34)

patch  1=( 43, 46, 45, 44)      Lower Ram Bottom Surface
patch  2=( 44, 45, 48, 47)
patch  3=( 45, 50, 49, 48)
patch  4=( 46, 51, 50, 45)

patch  5=( 52, 55, 54, 53)      Upper Ram Top Surface
patch  6=( 53, 54, 57, 56)
patch  7=( 54, 59, 58, 57)
patch  8=( 55, 60, 59, 54)

patch  9=( 47, 48, 82, 81)      Lower Ram Side Surface
patch 10=( 48, 49, 83, 82)
patch 11=( 49, 50, 84, 83)
patch 12=( 50, 51, 85, 84)

patch 13=( 96,  97, 57, 56)     Upper Ram Side Surface
patch 14=( 97,  98, 58, 57)
patch 15=( 98,  99, 59, 58)
patch 16=( 99,100, 60, 59)

patch 17=( 81, 82, 87, 86)      Felt Bottom Surface
patch 18=( 82, 83, 88, 87)
patch 19=( 83, 84, 89, 88)
patch 20=( 84, 85, 90, 89)

patch 21=( 96,  97,102,101)     Felt Top Surface
patch 22=( 97,  98,103,102)
patch 23=( 98,  99,104,103)
```

```
patch 24=( 99,100,105,104)

patch 25=( 86, 87,102,101)      Felt Side Surface
patch 26=( 87, 88,103,102)
patch 27=( 88, 89,104,103)
patch 28=( 89, 90,105,104)

patch 29=( 43,  47,  56, 52)    Central Surface
patch 30=( 43,  51,  60, 52)
patch 31=( 81,  96,101, 86)
patch 32=( 85,100,105, 90)

name=LRAMBS    patch=( 1, 2, 3, 4)

name=URAMTS    patch=( 5, 6, 7, 8)

name=RAMS      patch=( 9,10,11,12,13,14,15,16)

name=FELTS     patch=(17,18,19,20,21,22,23,24,
                      25,26,27,28)

name=CENTER    patch=(29,30,31,32)

name=MESH_CONTROL grid=(  2,   3,   4,  33,  34,  35,  36,  37,
                         66,  68,  70,  86,  88,  90,
                          1,   5,   9,  13,  17,  21,  25,  29,  43,  52,
                         111,112,113,114,115,116,117,118,119,
                         120,121,122,123,124,125,126,127,128)
```

Appendix C: Materials Property Library in Electroconsolidation

The files of materials property in MaPS property database library are listed as follows:

SiC.prop

```
/COM Silicon_Carbide Properties (SiC.prop)
/COM Preforms in electroconsolidation
MPTEMP
MPTEMP, 1,              25.0,
MPDATA,KXX , 1, 1,      75.0,
MPTEMP
MPTEMP, 1,              25.0,
MPDATA,DENS, 1, 1, 3217.0,
MPTEMP
MPTEMP, 1,          0.0,    100.0,    600.0,
MPDATA,C  , 1, 1, 598.3,    811.7,   1171.5,
```

YZ110HS.prop

```
/COM Norton Advanced Ceramics (YZ110HS.prop)
/COM Preforms in Electroconsolidation
MPTEMP
MPTEMP, 1,              25.0,
MPDATA,KXX , 1, 1,       2.0,
MPTEMP
MPTEMP, 1,              25.0,
MPDATA,DENS, 1, 1, 6070.0,
MPTEMP
MPTEMP, 1,          25.0,
MPDATA,C  , 1, 1, 580.0,
```

Powder-1.prop

```
/COM Graphite Carbon Powder (Powder-1.prop)
/COM medium at side of preform
MPTEMP
MPTEMP, 1,              25.0, 260.0, 538.0, 816.0, 1093.0, 1649.0,
MPDATA,KXX , 1, 1, 23.69, 15.40, 11.37, 8.29,  6.52,  4.27,
MPTEMP
MPTEMP, 1,              25.0,
```

```
MPDATA,DENS,  1,  1, 1485.7,
MPTEMP
MPTEMP,  1,          0.0,   226.8,   726.8, 1226.8, 1726.8, 2226.8, 2726.8,
MPDATA,C,     1,  1, 600.0, 1100.0, 1880.0, 2080.0, 2140.0, 2180.0, 2200.0,
```

Powder-2.prop

```
/COM  Graphite Carbon Powder (Powder-2.prop)
/COM  Bottom and top of medium (compaction 67.5%)
MPTEMP
MPTEMP,  1,          25.0, 260.0, 538.0, 816.0, 1093.0, 1649.0,
MPDATA,KXX ,  1,  1,  25.49, 16.56, 12.24,   8.92,    7.01,    4.59,
MPTEMP
MPTEMP,  1,          25.0,
MPDATA,DENS,  1,  1, 1540.7,
MPTEMP
MPTEMP,  1,          0.0,   226.8,   726.8, 1226.8, 1726.8, 2226.8, 2726.8,
MPDATA,C,     1,  1, 600.0, 1100.0, 1880.0, 2080.0, 2140.0, 2180.0, 2200.0,
```

---

die.prop

```
/COM  Graphite Die (die.prop)
/COM  Die chamber in Electroconsolidation
MPTEMP
MPTEMP,  1,          20.0, 230.0, 730.0, 1730.0, 2730.0,
MPDATA,KXX ,  1,  1,  90.0,   62.1,   32.9,    19.8,    15.8,
MPTEMP
MPTEMP,  1,          25.0,
MPDATA,DENS,  1,  1,  1720.0,
MPTEMP
MPTEMP,  1,          0.0,   226.8,   726.8, 1226.8, 1726.8, 2226.8, 2726.8,
MPDATA,C,     1,  1, 600.0, 1100.0, 1880.0, 2080.0, 2140.0, 2180.0, 2200.0,
```

---

ram.prop

```
/COM  Graphite Rams (ram.prop)
/COM  Ram chamber in Electroconsolidation
MPTEMP
MPTEMP,  1,          20.0, 230.0, 730.0, 1730.0, 2730.0,
MPDATA,KXX ,  1,  1,  90.0,   62.1,   32.9,    19.8,    15.8,
MPTEMP
```

```
MPTEMP, 1,                25.0,
MPDATA,DENS, 1, 1, 1840.0,
MPTEMP
MPTEMP, 1,              0.0,  226.8,  726.8, 1226.8, 1726.8, 2226.8, 2726.8,
MPDATA,C,    1, 1, 600.0, 1100.0, 1880.0, 2080.0, 2140.0, 2180.0, 2200.0,
```

---

filcarb.prop

```
/COM Fiber-Reinforced Sleeve Filcarb (filcarb.prop)
/COM filcarb material in Electroconsolidation
MPTEMP
MPTEMP, 1,                0.0, 250.0, 500.0, 1000.0, 1500.0, 2000.0,
MPDATA,KXX , 1, 1, 125.0,  96.3,  75.0,   50.0,   38.8,   32.5,
MPTEMP
MPTEMP, 1,                25.0,
MPDATA,DENS, 1, 1, 1650.0,
MPTEMP
MPTEMP, 1,              0.0,  226.8,  726.8, 1226.8, 1726.8, 2226.8, 2726.8,
MPDATA,C,    1, 1, 600.0, 1100.0, 1880.0, 2080.0, 2140.0, 2180.0, 2200.0,
```

---

felt.prop

```
/COM Felt Thermal Insulation (felt.prop)
/COM Thermal insulation outside of die in Electroconsolidation
MPTEMP
MPTEMP, 1,                0.0,  500.0, 1000.0, 1500.0, 2000.0, 2400.0,
MPDATA,KXX , 1, 1,   57.0,   57.1,   58.8,   58.9,   58.9,   59.1,
MPTEMP
MPTEMP, 1,                25.0,
MPDATA,DENS, 1, 1,  100.0,
MPTEMP
MPTEMP, 1,              25.0,  400.0,  800.0, 1200.0, 1600.0, 2000.0,
MPDATA,C  , 1, 1,  750.0, 1500.0, 1850.0, 1950.0, 2050.0, 2100.0,
```

---

constant_t.prop

```
/COM constant_t  air (constant_t.prop)
/COM Effective Constant Temperature
MPTEMP
MPTEMP, 1,                100.0,
MPDATA,DENS, 1, 1, 1.0e20,
```

```
MPTEMP
MPTEMP,  1,              100.0,
MPDATA,KXX ,  1,  1, 1.0e20,
MPTEMP
MPTEMP,  1,              100.0,
MPDATA,C   ,  1,  1, 1.0e20,
```

Distribution for ANL-00/33

Internal:

F. C. Chang (10)
T. H. Chien
H. M. Domanus
H. Drucker
A. M. Hassanein
C. A. Malefyt
R. B. Poeppel
R. C. Schmitt
R. A. Valentin
TIS Files

External:

DOE-OSTI (2)
Manager, DOE-Chicago
DOC Technology Administration
National Institute of Standards and Technology (NIST)
ANL Libraries
    ANL-E
    ANL-W
Energy Technology Division Review Committee:
    H. K. Birnbaum, University of Illinois, at Urbana-Champaign
    I.-W. Chen, University of Pennsylvania
    F. P. Ford, Rexford, NY
    S. L. Rhode, University of Nebraska - Lincoln
    H. S. Rosenbaum, Fremont, CA
    S. L. Sass, Cornell University
    R. Zoughi, University of Missouri-Rolla
Superior Graphite Company
    W. M. Goldberger (3)
    B. D. Merkle
    J. M. Borton
Biztek Consulting, Inc.
    R. R. Fessler