NASA Technical Memorandum 102556

COMGEN—A Computer Program for Generating Finite Element Models of Composite Materials at the Micro Level

Matthew E. Melis
Lewis Research Center
Cleveland, Ohio

April 1990

NASA

COMGEN - A COMPUTER PROGRAM FOR GENERATING FINITE ELEMENT

MODELS OF COMPOSITE MATERIALS AT THE MICRO LEVEL

Matthew E. Melis
National Aeronautics and Space Administration
Lewis Research Center
Cleveland, Ohio 44135

SUMMARY

COMGEN (COmposite Model GENerator) is an interactive FORTRAN program which can be used to create a wide variety of finite element models of continuous fiber composite materials at the micro level. It quickly generates batch or "session files" to be submitted to the finite element pre- and postprocessor PATRAN based on a few simple user inputs such as fiber diameter and percent fiber volume fraction of the composite to be analyzed. In addition, various mesh densities, boundary conditions, and loads can be assigned easily to the models within COMGEN. PATRAN uses a session file to generate finite element models and their associated loads which can then be translated to virtually any finite element analysis code such as NASTRAN or MARC.

INTRODUCTION

A significant percentage of time spent in a typical finite element analysis is taken up in the modeling and assignment of loads and constraints. This process not only requires the analyst to be well-versed in the art of finite element modeling, but also demands familiarity with some sort of preprocessing software in order to implement the task expediently.

VISCOA (VIScoplastic COmposite Analysis) (1), a program at the NASA Lewis Research Center to study the viscoplastic behavior of metal matrix composite materials using the finite element method, created a need for many types of models with various constraints and loadings for each. As a tool to assist in the generation of these models, COMGEN was written. COMGEN is an interactive user friendly FORTRAN computer program which prompts the user for pertinent geometric parameters, mesh densities, and loads based on the model which is selected, and then uses these inputs to assemble what is called a PATRAN session file.

PATRAN (2), a general purpose finite element pre- and postprocessor takes a session file generated by COMGEN and uses it to create a PATRAN data base which contains the geometric model along with the finite element connectivities, associated nodes, and the loadings. A session file is essentially a series of command lines in PATRAN syntax which are identical to those which a user would input interactively in PATRAN to create a given model. These files can be submitted to PATRAN in a batch mode in order to create a model data base in just a few minutes. If necessary, the model can be accessed to allow for any fine adjustments such as unusual loading conditions on the model which might be outside the scope of COMGEN. The model can then be translated from PATRAN to any analysis code the user desires through the use of interface translator programs which if not already available, can be easily written.

COMGEN performs the above tasks by tailoring previously written session files, which reside in its data base, to the parameters input during a COMGEN session. Additional models can be included in COMGENs library at any time by writing a PATRAN session file for the model and appending it to the COMGEN data base. It is important to note that additions to COMGEN do not necessarily have to be those of composite structures, but might be of any structure an analyst would desire to perform a number of parametric studies on. COMGEN supplies the platform to help with this task in a simple and expedient fashion. Appendix A gives more details on how to accomplish this.

The intentions of this paper are twofold: Its primary purpose is to serve as an instructional manual for both novice as well as experienced PATRAN users wishing to use COMGEN to assist in the modelling of composites. Secondly, but of no less importance, it demonstrates the usefulness of utilizing PATRAN session files in parametric studies and suggests how one might apply this theory to a wide variety of problems.

USING COMGEN

To study composites at the micro level with the finite element method, it is assumed that the material structure can be broken down into a "unit cell" of a fiber surrounded by matrix material. The initial version of COMGEN has two types of basic cells available from which to build composite models. The first is what is termed as a square packing arrangement where the fiber is positioned in the center of a square matrix cell. This is shown in figure 1. The second type, hexagonal packing, has the fiber centered in a hexagonal matrix cell shown in figure 2.

MODEL TYPES

The different models, or cases, in COMGEN are essentially all variations of square and hexagonal packing schemes. These variations include two- and three-dimensional cases, models with a fiber-matrix interface, and different constructions of the unit cells. Table B1 in appendix B shows a complete listing of the cases initially planned for entry into the COMGEN library. Figures B1 to B14 show the geometric arrangement of each of these cases as referenced from the table.

The following paragraphs describe in detail how to use the COMGEN program. In appendix A is an example of a typical COMGEN session which can be referred to as the reader progresses through the instructions to aid in their understanding. It should be noted that in this example, all of the user input for this session is identified by arrows along the left-hand side while brackets on the right-hand side group the various statements corresponding to the major headings below.

The first step in running COMGEN is to choose which model to create. The initial screen output in COMGEN displays the current models available. After entering the corresponding case number, COMGEN will echo the case number title and ask the user to verify that the correct case has been chosen. Next, COMGEN will ask for a name for the session file being created and then verifies that input. When COMGEN asks for verification of any input, a Y or N will appear in

brackets at the end of the sentence. This represents the yes or no default response to the question if only a carriage return is entered. If the user is working on a unix based system, it should be noted that unix is case sensitive. The file name can be entered in either case however the remainder of the COMGEN input should only be in capitals.

MODEL GEOMETRY

The next two parameters to be entered are the percent fiber volume fraction (FVR) and the fiber diameter. The FVR is entered in terms of the volume of the fibers in the composite divided by the total volume of the composite. The fiber diameter is entered in any dimensions the user chooses taking note to maintain consistency with these units throughout the remainder of the COMGEN session as well as with the material properties definitions later on.

If a case is selected where an interface between the fiber and matrix material exists, COMGEN will automatically prompt the user for the thickness of that interface. The interface will then be incorporated into the model while maintaining the diameter of the fiber entered. If the interface is thin compared to the overall model, caution should be used in interpreting the results of the analysis due to potentially poor aspect ratios of the elements in the interface.

Choosing a two-dimensional case in COMGEN gives the analyst the option of using either a plane strain or plane stress element. This prompt is also automatic if appropriate. A three-dimensional model at this time will only use an eight-node hex element. If a three-dimensional model is selected, COMGEN will ask the user if a cubic model is desired. Entering Y (carriage return) to this would result in a model with its depth being exactly equal to its width. The depth or thickness of the model can be entered manually if the dimension is to be otherwise. Note from the appendix figures that the hexagonal cases do not have equal width and height dimensions thus the models cannot be cubic as with the square models.

MESH DENSITY

PATRAN models are initially described geometrically with two- and three-dimensional regions called patches and hyperpatches respectively. The nodal density in each of these regions is determined by the number of nodes the user assigns along their edges. Once the model characteristics are defined in COMGEN, the user must decide upon the nodal density of the mesh. This is done with the help of figures 3 to 5. Each of these figures show the unique geometric regions that combine to make up the models. The figures apply to both the two- and three-dimensional cases with the three-dimensional cases simply having a thickness associated with them out of the paper.

Figure 3 depicts the regions that make up any model based on a square packing scheme for FVRs of up to 20 percent. If the FVR is greater than 20 percent for square packing, the geometry changes such that only four unique regions make up the model in order to accommodate a larger fiber in the model. This is shown in figure 4. The hexagonal packing cases are configured from three redundant regions for any FVR to define their shape as seen in figure 5.

3

To define the density of nodes in a model, COMGEN requests the number of nodes along the edges of the regions labeled in figures 3 to 5 based on the type of model selected. Although not shown in the above figures, if the model has a fiber-matrix interface, COMGEN will ask for the number of nodes through its thickness. When all of the directions have been defined, COMGEN prints out the values and gives the option to redefine them before continuing on if necessary.

MATERIAL DEFINITIONS

Presently, COMGEN does not generate or write any material property definitions to the session files. This is left up to the user to either add once inside PATRAN or to insert manually to the analysis code data deck. COMGEN does however assign different property numbers to the fiber, interface, and matrix elements to help distinguish them when assigning the material properties later on.

MODEL OPTIMIZATION

After defining the mesh density, the user has a choice of having PATRAN optimize the model after its construction. Having PATRAN do this takes a significant amount of time for larger models and can most likely be done by the analysis code being used so it is strictly up to the user whether to invoke this option or not.

LOADS AND CONSTRAINTS

The final modeling step in COMGEN is an option to add simple loading and/ or constraints to the model in the form of pressures, forces, displacements, and temperatures. If the user selects to add one or more of the above, COMGEN will first ask for the type of loading using the four letter acronyms: PRES, FORC, TEMP, or DISP. COMGEN will then ask for the plane which is to be loaded. This can be determined by referring to figure 6 if the model is two dimensional, or figure 7 if the model is three dimensional. These figures show the orientation of the models with respect to the global axes and have each of their planes numbered. After choosing the plane number, COMGEN prompts for the set ID to be assigned to the load. This is used if the user wishes to have several loading cases in one analysis. Any integer can be used for this ID and there is no default for it.

If FORC, TEMP, or DISP are chosen as loads, the user is asked to enter the magnitude and direction of the load in each of six directions with respect to the global axes defined in figures 6 and 7. These values are entered for the X, Y, and Z directions and the X, Y, and Z rotations respectively with each value being separated by slashes (/). This format is identical to that for PATRAN and is written directly to the session file as entered by the user. It is important to note for displacement data that a zero and a blank are not interchangeable. A blank simply indicates no input, a zero however indicates that there is zero displacement of the nodes on the selected plane in the direction associated with the data field.

4

Due to the way that PATRAN handles the assignment of pressures, a slightly different approach to entering the data must be used when selecting the PRES option. After defining the load set ID and which plane is to be loaded, COMGEN will individually prompt the user for the magnitude of the pressure in each of the six directions relative to the global axis. Despite the fact that COMGEN will write pressures to any direction, it is highly recommended that ONLY normal pressures are assigned to a given face. COMGEN is only verified for these normal pressures and many problems can arise in the analysis using pressures not perpendicular to the surface of the model. It is also recommended that the user verify these loads in PATRAN to avoid any possible loading errors.

After completing a load assignment COMGEN asks the user if any other loads are to be added to the model. An affirmative response to this will start the above process again and shall continue as such until all the loading assignments are complete.

INTERFACING WITH PATRAN

To submit a COMGEN session file to PATRAN interactively, simply call PATRAN, specify the terminal device number, and type SES instead of GO when prompted for. PATRAN will ask for the session file name and upon receiving it will proceed automatically with the creation of the model. The graphics are turned off in all COMGEN session files in order to decrease the generation time of the model so there will be no graphics seen on the screen until the model is complete and the graphics are automatically turned on again. If it is desired to watch the model construction take place, edit the session file before PATRAN submission and remove the line "SET,GRAPHICS,OFF" and then five lines below this input the three lines "WI," "4," and "7" in that order. Submit the session file to PATRAN as described above and the model generation can be watched on the screen.

Once the PATRAN data base is built and any final adjustments made, it has to be converted to a PATRAN neutral file. It is then translated to a specific analysis code format via an interface program. For the novice user there is an on line help system within PATRAN itself. It can be accessed by simply typing help and then stepping through the menus shown on the screen. If so desired, one can move directly to a specific help screen if the proper keyword is known. For example if a user wants to know more on how to change the view of a model, the command "HELP,VIEW" would go directly to the help screen for this option as opposed to stepping down through the menus. Below are some useful keywords and their meanings.

KEYWORD	MEANING
WINDOW	WINDOWING COMMANDS
VERIFY	MODEL VERIFICATION OPTIONS
PLOT	PLOTTING OPTIONS
FORC	FORCE LOAD OPTIONS
PRES	PRESSURE LOAD OPTIONS
DISPL	DISPLACEMENT OPTIONS
TEMP	TEMPERATURE OPTIONS

CONCLUDING REMARKS

At the time of publication, COMGEN and its data base consisted of over 3000 lines of FORTRAN code thus making it unrealistic to incorporate the source into this paper. It is available upon request from the author at any time.

It is anticipated that COMGEN will be a useful tool to the novice as well as the experienced users of PATRAN in generating finite element models of composites as well as for any other new additions which one might wish to include. The preceding pages show that with only a minimum amount of time and effort, a relatively complicated finite element model can be quickly generated with COMGEN thus significantly reducing the analysts work load in the analysis process.

REFERENCES

1. VISCOA (VIScoplastic COmposite Analysis) Research program at Lewis Research Center to investigate viscoplastic material behavior of metal matrix composites. Contact Author for further information.

2. PATRAN II Users Manual, PDA Engineering, Santa Ana, CA 1984.

APPENDIX A - COMGEN SESSION EXAMPLE

   The following example shows the COMGEN session which generated a model with
its loads and constraints for the one cell case shown in two separate views in
figures Al(a) and (b). Arbitrary loads were chosen to simply show how they are
added in COMGEN. Arrows indicate user input. Comments for clarification are
made inside parenthesis.

```
           WELCOME TO THE COMGEN COMPOSITE MODELER                    ┐

        ****************************************************
        ****************************************************
        **                                              **
        **     CASES PRESENTLY AVAILABLE IN COMGEN       **
        **                                              **
        ****************************************************
        ****************************************************
        **                                              **
        **                                              **
        **     CASE 1 .... 2-D ONE CELL SQUARE           **
        **     CASE 2 .... 2-D NINE CELL SQUARE          **
        **     CASE 3 .... 2-D NINE CELL REDUCED SQUARE  **
        **     CASE 4 .... 3-D ONE CELL SQUARE           **
        **     CASE 5 .... 3-D NINE CELL SQUARE          **
        **     CASE 6 .... 3-D NINE CELL REDUCED SQUARE  **
        **     CASE 7 .... 2-D ONE CELL SQUARE W/ INTF   **
        **     CASE 10 ... 3-D ONE CELL SQUARE W/ INTF   **
        **     CASE 13 ... 2-D ONE CELL HEX              **
        **     CASE 14 ... 2-D SEVEN CELL HEX            **
        **     CASE 15 ... 2-D SEVEN CELL REDUCED HEX    **
        **     CASE 16 ... 2-D SEVEN CELL EXPANDED HEX   **
        **     CASE 17 ... 3-D ONE CELL HEX              **
        **     CASE 18 ... 3-D SEVEN CELL HEX            **
        **     CASE 19 ... 3-D SEVEN CELL REDUCED HEX    **
        **     CASE 20 ... 3-D SEVEN CELL EXPANDED HEX   **
        **                                              **
        ****************************************************
        ****************************************************


        !!!!   PLEASE ENTER ALL DATA IN CAPITALS   !!!!
        ENTER THE GEOMETRY CASE NUMBER (1-28)
----->  4
        ONE CELL SQUARE, 3D GEOMETRY SELECTED
        IS THIS THE CORRECT CASE? [Y]
----->  Y
        ENTER THE FILENAME RECEIVING THE SESSION FILE DATA:
        THE SIZE IS LIMITED TO TEN CHARACTERS.
----->  case4
        THE FILENAME YOU ENTERED IS case4   , CORRECT? [Y]
----->  Y
        PLEASE ENTER THE FIBER VOLUME RATIO:
----->  .1
        PLEASE ENTER THE FIBER DIAMETER:
----->  .08
        DO YOU WANT A CUBIC MODEL? [Y]
----->  N
        PLEASE ENTER THE THICKNESS OF THE MODEL:
----->  .1
        ENTER NUMBER OF NODES THROUGH THE THICKNESS.
----->  2
```

MODEL SELECTION

MODEL GEOMETRY

8

```
        NODES THROUGH THE THICKNESS =              2
        DEFINE NUMBER OF NODES IN THE FOLLOWING DIRECTIONS
        FOR THE GEOMETRIC REGIONS OF THE SQUARE MODEL WITH
        AN FVR < OR =.2 AS DESCRIBED IN THE USERS MANUAL
        B:
----->  2
        C:
----->  3
        D:
----->  4
        E:
----->  5
        5=             2 C=          3 D=          4 E=          5
        ARE THESE VALUES CORRECT ?  [Y]
----->  Y
        DO YOU WANT THE MODEL OPTIMIZED? (Y/N) [N]
----->  N
        LOADING SPECIFICATION SECTION
        DO YOU WANT TO ADD LOADS? [Y]
----->  Y
        ENTER THE TYPE OF LOAD (PRES,FORC,DISP,TEMP)
----->  PRES (Pressure load of 1.0 in the minus z direction)
        WHAT PLANE IS TO BE LOADED?
----->  1
        ENTER LOAD SET ID NUMBER:
----->  1

  ***********************************************************
  ***********************************************************
  **                                                     **
  **   USING THE GLOBAL AXES, PRESSURES CAN BE ENTERED IN SIX  **
  **   DIFFERENT DIRECTIONS. ENTER THE DESIRED DATA WHEN  **
  **   PROMPTED FOR BELOW.  IT IS RECOMMENDED THAT ONLY NORMAL **
  **   PRESSURES ARE USED FOR ANY OF THE MODELS AT THIS TIME.  **
  **   DUE TO THE WAY PATRAN ASSIGNS THESE VALUES.  FORCES  **
  **   AND DISPLACEMENTS CAN SAFELY BE ENTERED IN ANY FASHION  **
  **   DESIRED. BE SURE TO ENTER ZERO FOR THE PRESSURE   **
  **   DIRECTIONS HAVING NO DATA.                        **
  **                                                     **
  ***********************************************************
  ***********************************************************

        X DIRECTION
----->  0.0
        Y DIRECTION
----->  0.0
        Z DIRECTION
----->  -1.0
        X ROTATION
----->  0.0
        Y ROTATION
```

```
----> 0.0
      Z ROTATION
----> 0.0
      DO YOU WANT TO ADD MORE LOADS? (Y/N) [Y]
----> Y
      ENTER THE TYPE OF LOAD (PRES,FORC,DISP,TEMP)
----> PRES (pressure load of 2.0 in positive z)
      WHAT PLANE IS TO BE LOADED?
----> 2
      ENTER LOAD SET ID NUMBER:
----> 1

      **************************************************************
      **************************************************************
      **                                                        **
      **  USING THE GLOBAL AXES, PRESSURES CAN BE ENTERED IN SIX **
      **  DIFFERENT DIRECTIONS. ENTER THE DESIRED DATA WHEN      **
      **  PROMPTED FOR BELOW.  IT IS RECOMMENDED THAT ONLY NORMAL **
      **  PRESSURES ARE USED FOR ANY OF THE MODELS AT THIS TIME. **
      **  DUE TO THE WAY PATRAN ASSIGNS THESE VALUES.  FORCES    **
      **  AND DISPLACEMENTS CAN SAFELY BE ENTERED IN ANY FASHION **
      **  DESIRED. BE SURE TO ENTER ZERO FOR THE PRESSURE        **
      **  DIRECTIONS HAVING NO DATA.                             **
      **                                                        **
      **************************************************************
      **************************************************************

      X DIRECTION
----> 0.0
      Y DIRECTION
----> 0.0
      Z DIRECTION
----> 2.0
      X ROTATION
----> 0.0
      Y ROTATION
----> 0.0
      Z ROTATION
----> 0.0
      DO YOU WANT TO ADD MORE LOADS? (Y/N) [Y]
----> Y
      ENTER THE TYPE OF LOAD (PRES,FORC,DISP,TEMP)
----> FORC (Force load of 3.0 in Y)
      WHAT PLANE IS TO BE LOADED?
----> 3
      ENTER LOAD SET ID NUMBER:
----> 1
      ENTER THE MAGNITUDE OF THE LOADS IN GLOBAL AXES,
      BASED ON PATRAN DEFEG CONVENTIONS SHOWN BELOW
      EX. X/Y/Z/RX/RY/RZ
      IF A BLANK IS DESIRED, ENTER IT AS SUCH.
      EX.  X//Z/RX/RY//  MAX 20 CHARACTERS
----> /3
      DO YOU WANT TO ADD MORE LOADS? (Y/N) [Y]
----> Y
```

LOADS AND CONSTRAINTS

10

```
          ENTER THE TYPE OF LOAD (PRES,FORC,DISP,TEMP)
----->    FORC (Force load of minus 4.0 in X)
          WHAT PLANE IS TO BE LOADED?
----->    4
          ENTER LOAD SET ID NUMBER:
----->    1
          ENTER THE MAGNITUDE OF THE LOADS IN GLOBAL AXES,
          BASED ON PATRAN DEFEG CONVENTIONS SHOWN BELOW
          EX. X/Y/Z/RX/RY/RZ
          IF A BLANK IS DESIRED, ENTER IT AS SUCH.
          EX.  X//Z/RX/RY//  MAX 20 CHARACTERS
----->    -4
          DO YOU WANT TO ADD MORE LOADS? (Y/N) [Y]
----->    Y
          ENTER THE TYPE OF LOAD (PRES,FORC,DISP,TEMP)
----->    DISP (Zero displacement in Y)
          WHAT PLANE IS TO BE LOADED?
----->    5
          ENTER LOAD SET ID NUMBER:
----->    1
          ENTER THE MAGNITUDE OF THE LOADS IN GLOBAL AXES,
          BASED ON PATRAN DEFEG CONVENTIONS SHOWN BELOW
          EX. X/Y/Z/RX/RY/RZ
          IF A BLANK IS DESIRED, ENTER IT AS SUCH.
          EX.  X//Z/RX/RY//  MAX 20 CHARACTERS
----->    /0///
          DO YOU WANT TO ADD MORE LOADS? (Y/N) [Y]
----->    Y
          ENTER THE TYPE OF LOAD (PRES,FORC,DISP,TEMP)
----->    TEMP (Temperature load of 100)
          WHAT PLANE IS TO BE LOADED?
----->    6
          ENTER LOAD SET ID NUMBER:
----->    2
          ENTER THE MAGNITUDE OF THE LOADS IN GLOBAL AXES,
          BASED ON PATRAN DEFEG CONVENTIONS SHOWN BELOW
          EX. X/Y/Z/RX/RY/RZ
          IF A BLANK IS DESIRED, ENTER IT AS SUCH.
          EX.  X//Z/RX/RY//  MAX 20 CHARACTERS
----->    100
          DO YOU WANT TO ADD MORE LOADS? (Y/N) [Y]
----->    N
          SESSION FILE case.c4    IS ASSEMBLED FOR PATRAN
```

LOADS AND CONSTRAINTS

APPENDIX B

Figures B1 to B14 depict the basic geometric layout for each of the cases listed in table B1. referring to these figures can help the user decide which case to select for a particular analysis.

COMGEN STRUCTURE

The basic foundation of which COMGEN is built upon is very simple. COM-GENs primary function is to modify a previously written PATRAN session file for a given model from the data base "SESSION.DAT" such that it will build a PATRAN model based on geometric parameters entered in COMGEN by the user. When the initial session file is written for the SESSION.DAT data base, all of the pertinent dimensional data as well as nodal density data in the PATRAN command lines, are written in as variable names. These names are then assigned values at the beginning of the session with a series of PATRAN "define" statements. COMGEN computes these values based on the users input and then writes the necessary define statements at the beginning of the session file being assembled. Loading and optimization statements are added at the end of this file before closing it if opted for.

The PATRAN loading and constraint statements reside in COMGEN as opposed to SESSION.DAT to allow for the needed flexibility in their assignment to the model. COMGEN recognizes each model case as being unique and formulates the load statements as such. A simplified flow chart of the COMGEN process is shown in figure B15.

ADDING CASES TO COMGEN

Incorporating new models into COMGEN is relatively easy to do but requires a strong working knowledge of PATRAN hence the "non-PATRAN user will not understand the following procedure. This process can be broken down into these six steps:

1. The first step is to create a basic session file which will build the new model. Care must be taken to use only defined variables in place of any numerical parameters which the user will wish to vary. With all of the required "define statements" at the top of the file, run the session to see if it works. If desired, include the equivalencing routines and the PMAT and PFEG cards, etc. After debugging the file, remove the define statements, and the PATRAN start up and stop commands, and add an eight line header to the beginning of the file. The existing session files in the data base can be used as examples.

2. Load the completed session file at the end of the session data base SESSION.DAT taking note of the starting and ending line numbers of the new addition.

3. Access COMGEN at the beginning of the program and assign the starting line number to the array CASES(GCASE,1) and the ending line number in CASES(GCASE,2) where GCASE is a case number chosen for the new model. Add this case as available in the print screen of "CASES PRESENTLY AVAILABLE."

4. Following the request for the case number GCASE, are a series of if statements which echo the chosen case number back to the user. Enter a set of similar statements for your case. Note that the variable KASE is used to differentiate between high and low FVR problems of the same case. Be sure to take this into account if there are variations of the model based on FVR.

5. Go to the ENDIF following statement 8. If more geometric information is required other than what COMGEN presently asks for, write in the necessary statements to acquire that data from the user.

6. Continue down through the "case selection section" and write an ELSEIF loop to include your case number. Within this loop subroutines must be called to formulate and write all of the statements to be added to the file being assembled. Follow the example of the existing subroutines to better understand this process and to help write the required subroutines for the new model. Subroutines HEXCAL and SQCAL compute the geometric variables and write out the associated define statements for the hexagonal and square packed models respectively. The other subroutines are straight forward and can be easily understood with a little effort. The addition of the new model should now be complete.
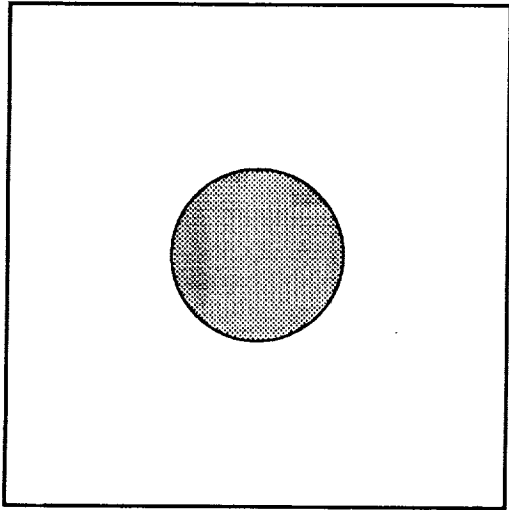
Figure 1. - Square cell packing arrangement.
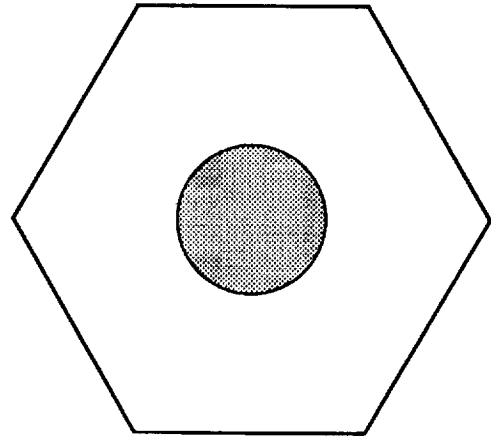
Figure 2. - Hexagonal cell packing arrangement.
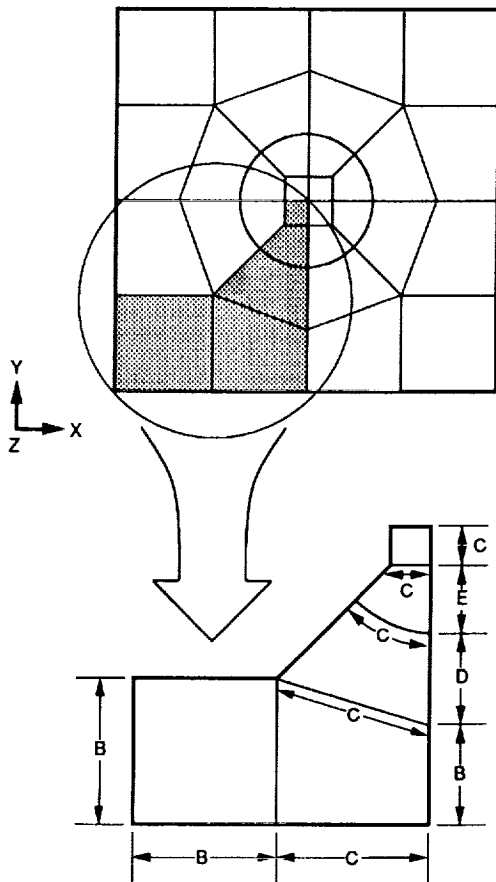
Figure 3. - Labels corresponding to COMGEN nodal density assignments for square packing arrangement, FUR ≤ 20 %.
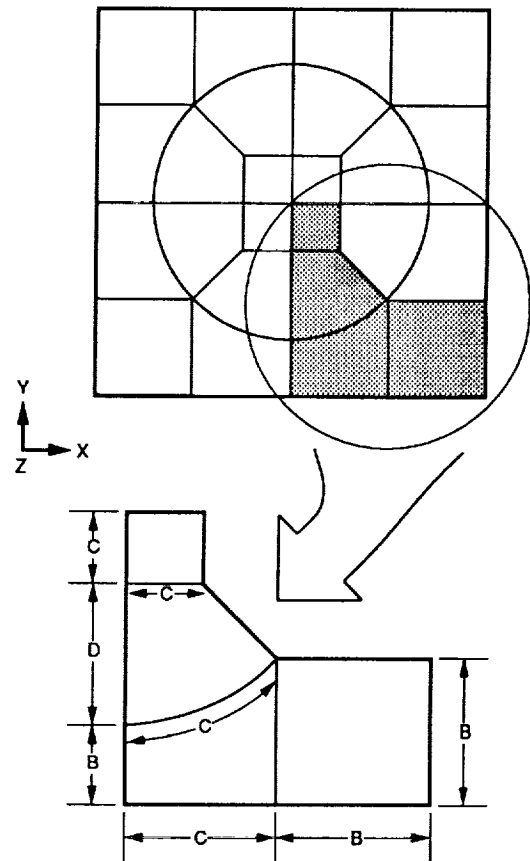
Figure 4. - Labels corresponding to COMGEN nodal density assignments for square packing arrangement, FUR > 20 %.

14
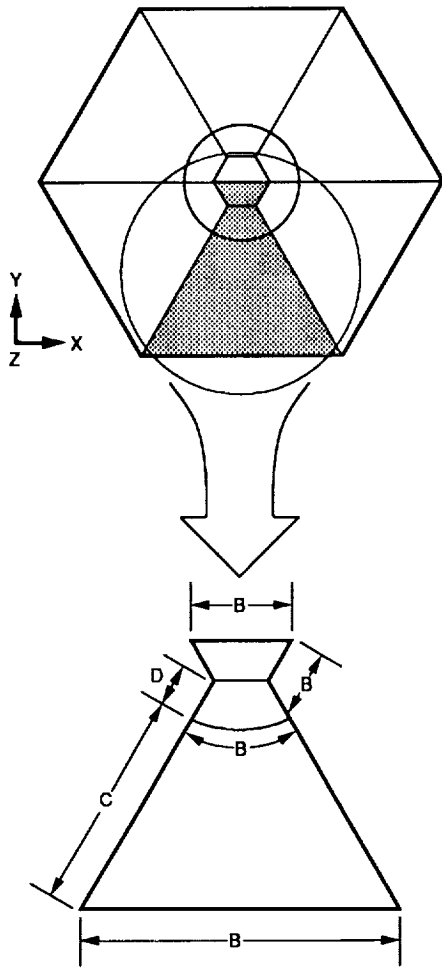
Figure 5. - Labels corresponding to COMGEN nodal density assignments for hexagonal packing arrangement.
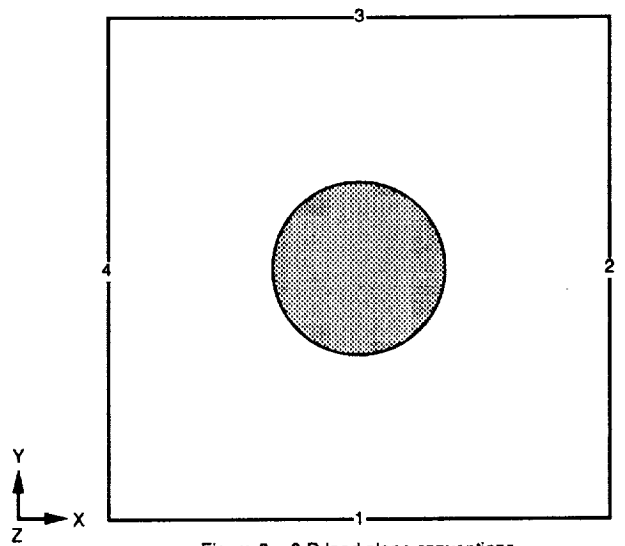
Figure 6. - 2-D load plane conventions.
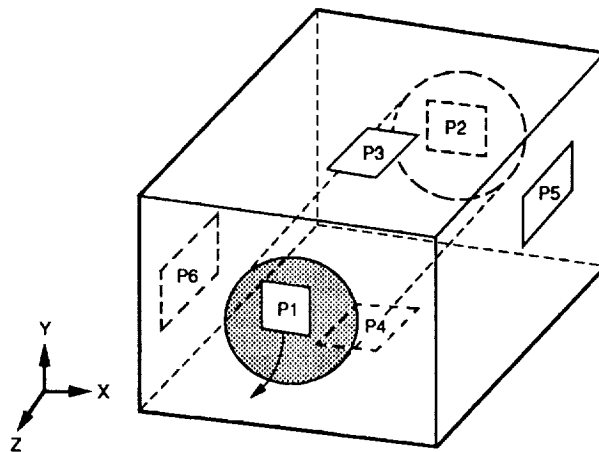
Figure 7. - 3-D load plane conventions.

15

(a) Front view.
(b) Side view.
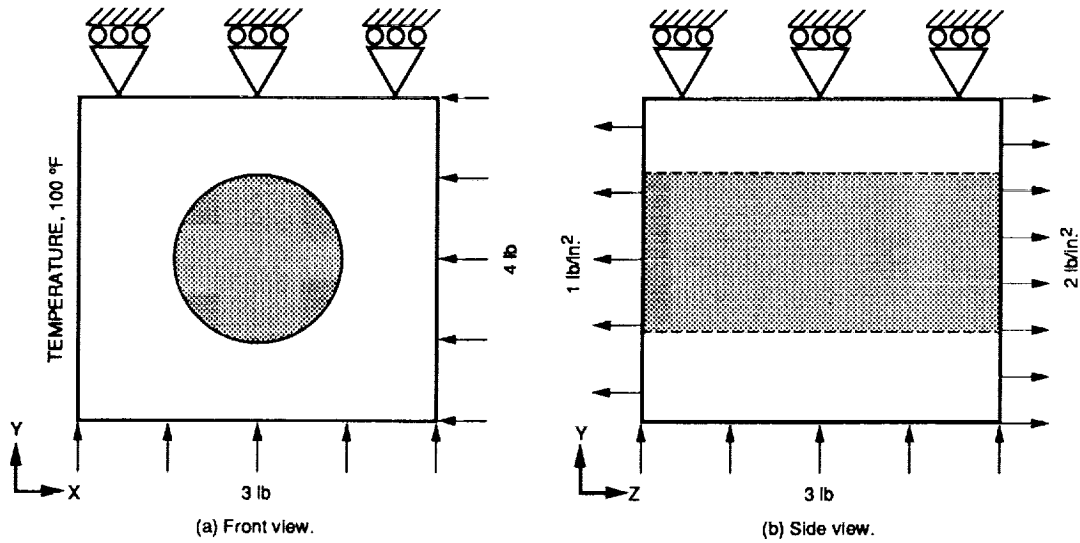
Figure A1. - Loading conditions for sample test case.

TABLE B1

Case number	Model type	Packing type	Interface	Figure
Case 1	2-D one cell	Square	No	B1
Case 2	2-D nine cell	Square	No	B2
Case 3	2-D reduced nine cell	Square	No	B3
Case 4	3-D one cell	Square	No	B1
Case 5	3-D nine cell	Square	No	B2
Case 6	3-D reduced nine cell	Square	No	B3
Case 7	2-D one cell	Square	Yes	B4
Case 8	2-D nine cell	Square	Yes	B5
Case 9	2-D reduced nine cell	Square	Yes	B6
Case 10	3-D one cell	Square	Yes	B4
Case 11	3-D nine cell	Square	Yes	B5
Case 12	3-D reduced nine cell	Square	Yes	B6
Case 13	2-D one cell	Hexagonal	No	B7
Case 14	2-D seven cell	Hexagonal	No	B8
Case 15	2-D reduced seven cell	Hexagonal	No	B9
Case 16	2-D expanded seven cell	Hexagonal	No	B10
Case 17	3-D one cell	Hexagonal	No	B7
Case 18	3-D seven cell	Hexagonal	No	B8
Case 19	3-D reduced seven cell	Hexagonal	No	B9
Case 20	3-D expanded seven cell	Hexagonal	No	B10
Case 21	2-D one cell	Hexagonal	Yes	B11
Case 22	2-D seven cell	Hexagonal	Yes	B12
Case 23	2-D reduced seven cell	Hexagonal	Yes	B13
Case 24	2-D expanded seven cell	Hexagonal	Yes	B14
Case 25	3-D one cell	Hexagonal	Yes	B11
Case 26	3-D seven cell	Hexagonal	Yes	B12
Case 27	3-D reduced seven cell	Hexagonal	Yes	B13
Case 28	3-D expanded seven cell	Hexagonal	Yes	B14

16
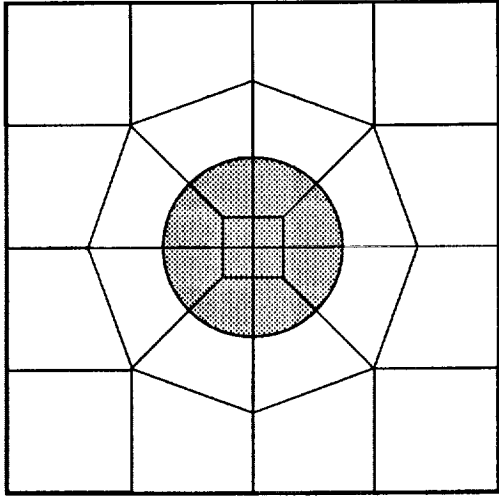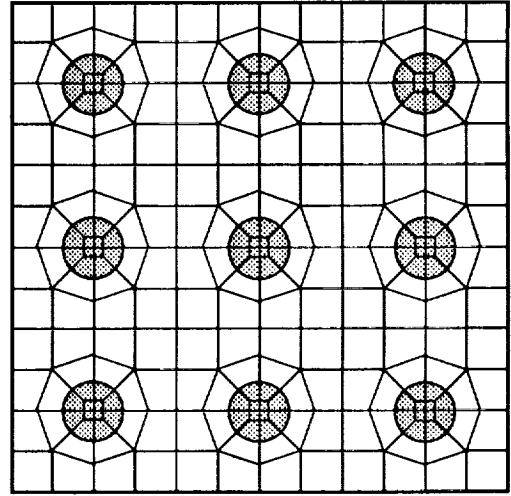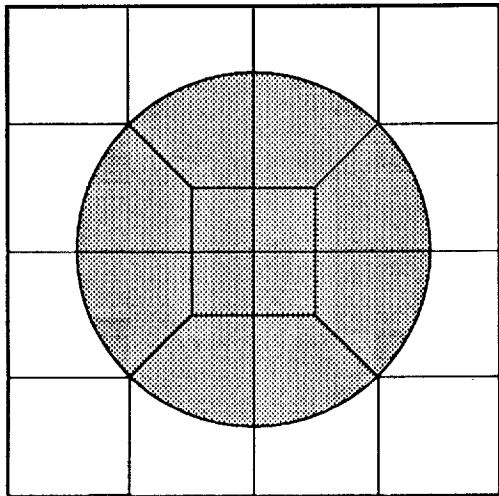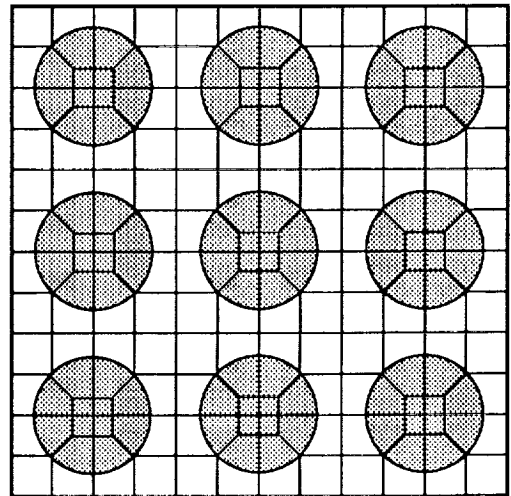
(a) FUR ≤ 20 %.

(b) FUR > 20 %.

Figure B1. - Geometric layout of one cell square packing
scheme.

(a) FUR ≤ 20 %.

(b) FUR > 20 %.

Figure B2. - Geometric layout of nine cell square packing
scheme.

17
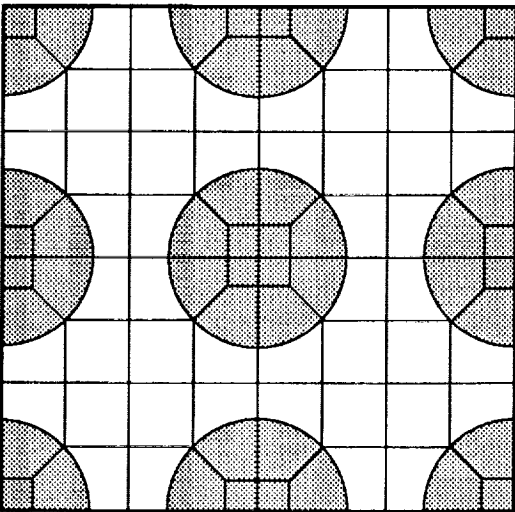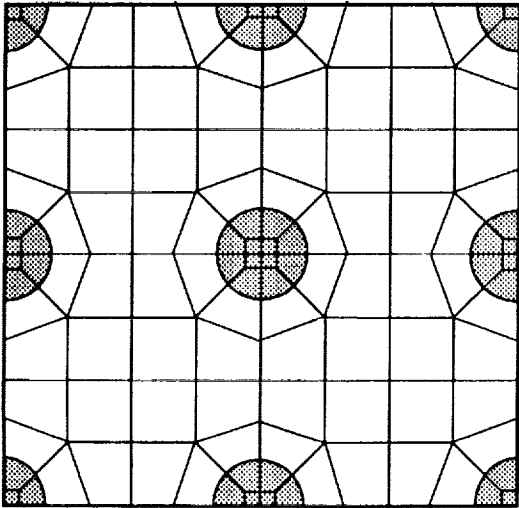
Figure B3. - Geometric layout for reduced nine cell square packing scheme.
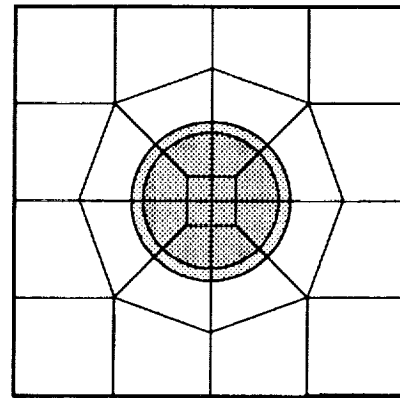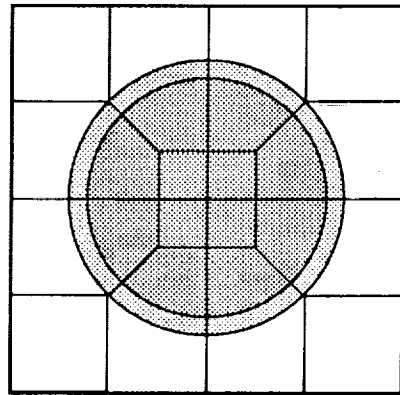
(a) FUR ≤ 20 %.

(b) FUR > 20 %.

Figure B4. - Geometric layout of one cell square packing scheme with interface layer.

18
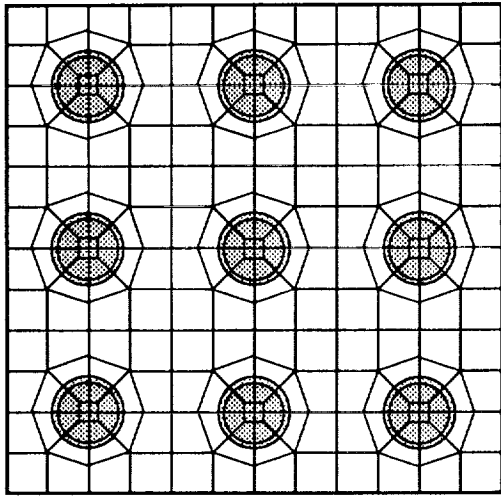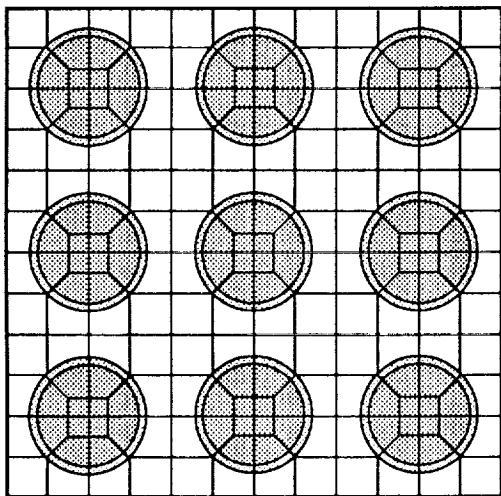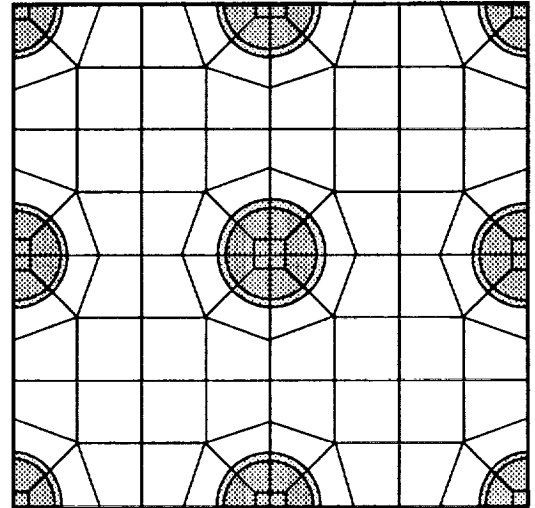
(a) FUR ≤ 20 %.

(b) FUR > 20 %.
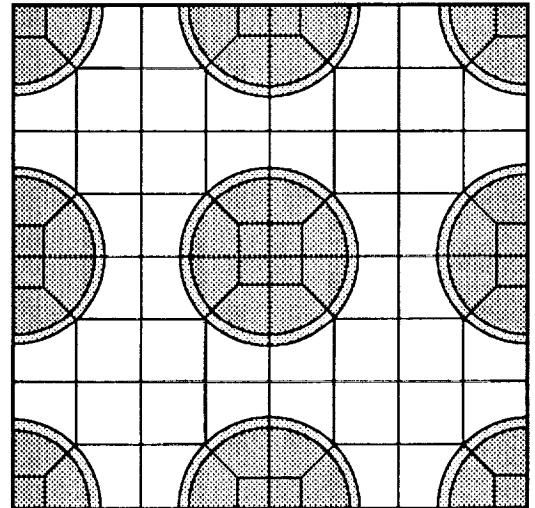
Figure B5. - Geometric layout of nine cell square packing scheme with interface layer.

Figure B6. - Geometric layout for reduced nine cell square packing scheme with interface layer.

19
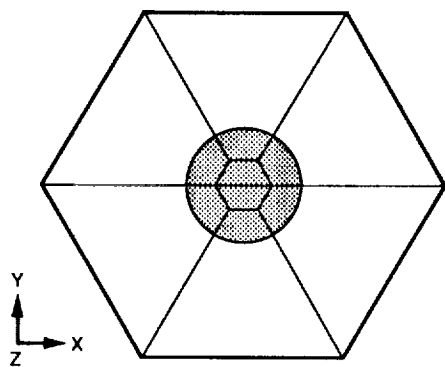
Figure B7. - Geometric layout for one cell hexagonal
packing scheme.
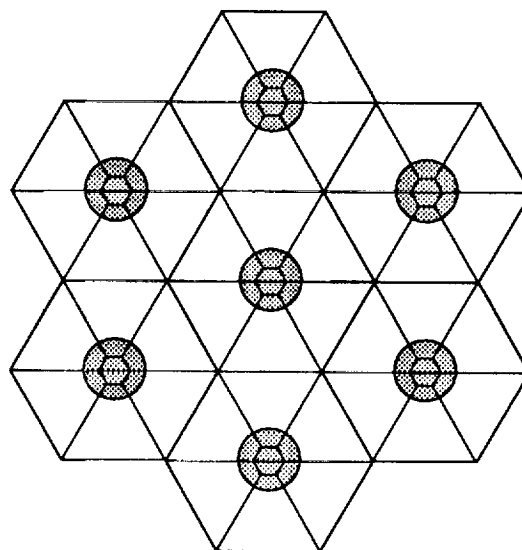
Figure B8. - Geometric layout for a 7 cell hexagonal packing
scheme.
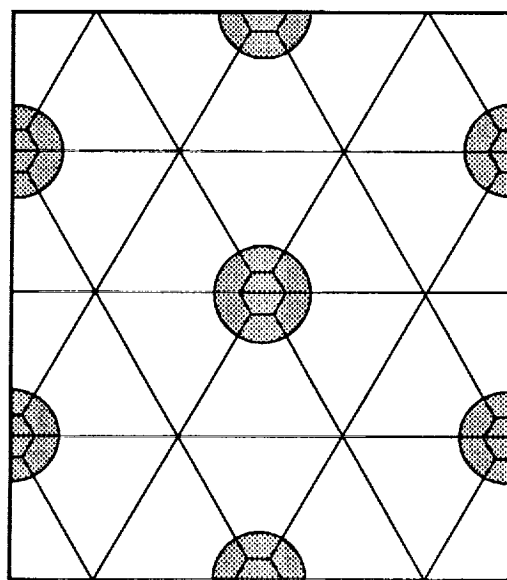
Figure B9. - Geometric layout for a reduced 7 cell hexagonal
packing scheme.

20
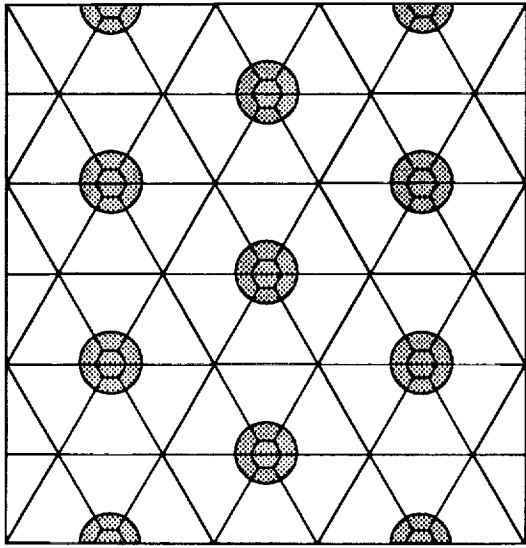
Figure B10. - Geometric layout for an expanded 7 cell hexagonal packing scheme.
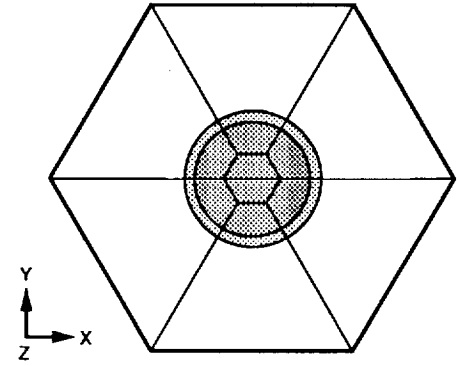
Figure B11. - Geometric layout for one cell hexagonal packing scheme.
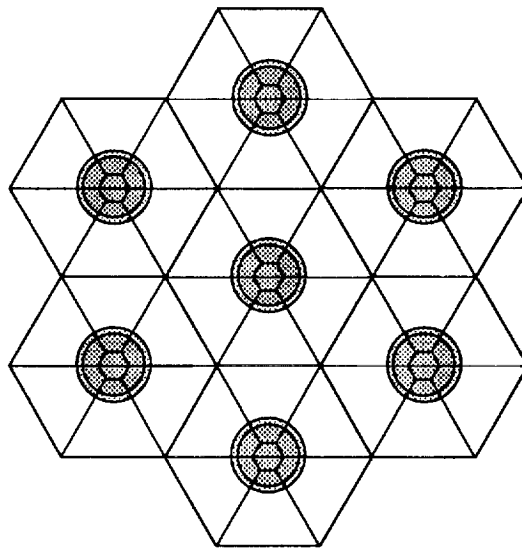
Figure B12. - Geometric layout for a 7 cell hexagonal packing scheme.

21
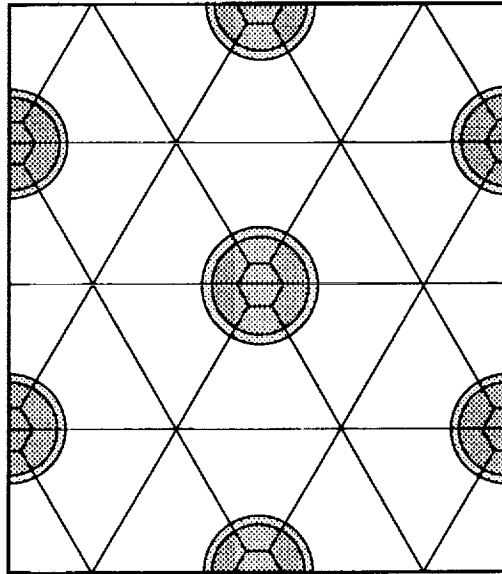
Figure B13. - Geometric layout for a reduced 7 cell hexagonal
packing scheme.
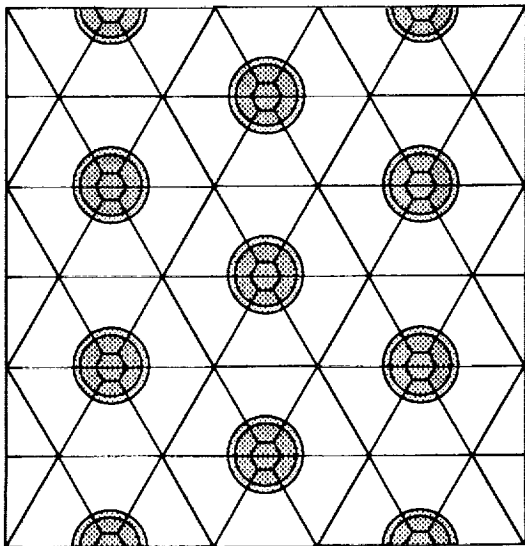
Figure B14. - Geometric layout for an expanded 7 cell hexagonal
packing scheme.
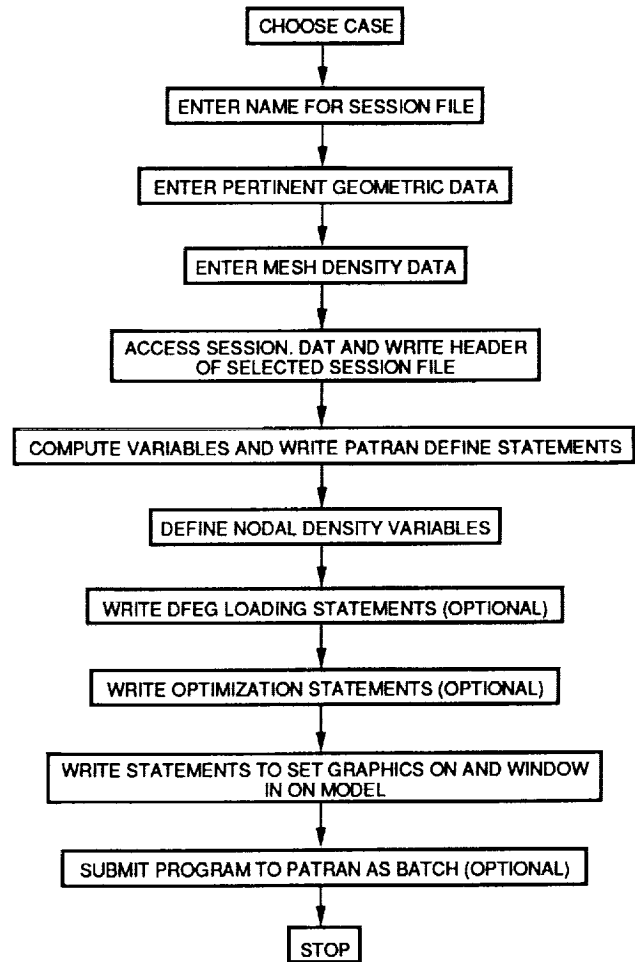
Figure B15. - COMGEN Methodology.

22

16. Abstract

COMGEN (COmposite Model Generator) is an interactive FORTRAN program which can be used to create a wide variety of finite element models of continuous fiber composite materials at the micro level. It quickly generates batch or "session files" to be submitted to the finite element pre- and postprocessor PATRAN based on a few simple user inputs such as fiber diameter and percent fiber volume fraction of the composite to be analyzed. In addition, various mesh densities, boundary conditions, and loads can be assigned easily to the models within COMGEN. PATRAN uses a session file to generate finite element models and their associated loads which can then be translated to virtually any finite element analysis code such as NASTRAN or MARC.