COMGEN-BEM: BOUNDARY ELEMENT MODEL GENERATION FOR COMPOSITE MATERIALS MICROMECHANICAL ANALYSIS

Robert K. Goldberg
NASA Lewis Research Center
Cleveland, OH 44135

and

Michele D. Comiskey*
University of Akron
Akron, OH 44309

ABSTRACT

COMGEN-BEM (Composite Model Generation - Boundary Element Method) is a program developed in MSC/PATRAN's PATRAN Command Language (PCL) which generates boundary element models of continuous fiber reinforced laminated and woven composites at the micromechanical (constituent) scale. Through the use of menus and forms, the user enters a few simple parameters such as fiber volume fraction, fiber diameter, mesh density, material properties, fiber rotation information and load and boundary condition data. From the user defined parameters, a complete boundary element model is automatically generated. Once the model is generated, the user can invoke a provided translator to convert the model information into an appropriate boundary element analysis input format. This program demonstrates the ability of MSC/PATRAN and PCL to simplify the parametric generation of boundary element models in general, and composite micromechanical models in particular.

---

*NASA/OAI Collaborative Aerospace Internship and Fellowship Program

INTRODUCTION

In the development of the next generation of aircraft engines, composite materials appear to combine superior mechanical properties with the low weight required for operation in the required high temperature environments. To make optimal use of these materials, advanced analytical methods must be developed which allow the computation of composite effective properties as well as detailed interior stress distributions. A joint program between the State University of New York at Buffalo and NASA Lewis has been underway for several years to examine the possible application of the boundary element method (BEM) to the analysis of composite materials at the micromechanical (constituent) scale. The motivation behind the use of the boundary element method is the ability of BEM to model a three-dimensional structure using surface discretization only.

Much of the time and effort involved in conducting a boundary element analysis is the construction of the boundary element model. Particularly, if a parametric study is conducted in which parameters such as fiber diameter, mesh density or loading conditions are varied, the effort involved in constructing multiple models can be significant. To simplify this process, the program COMGEN-BEM (Composite Model Generation - Boundary Element Method) was developed. This program, which is written in MSC/PATRAN's PATRAN Command Language (PCL) [1], generates boundary element models of fiber-reinforced laminated and plain weave woven composite materials at the micromechanical level based on geometric, mesh density, fiber rotation, material property and load and boundary condition information input by the user. The user input is conducted through the use of menus and forms. In this manner, the generation of the boundary element model is significantly simplified, and the user does not need to have a detailed knowledge of either MSC/PATRAN or boundary element modeling.

COMGEN-BEM is based on the FORTRAN program COMGEN [2], which generated MSC/PATRAN Version 2.5 compatible session files based on user input to allow for the creation of finite element models of composite materials at the micromechanical level. While some of the basic structure and formulation of COMGEN-BEM is taken directly from COMGEN, the two programs differ in several significant aspects. First, while COMGEN was designed to simplify the creation of finite element models of composites, COMGEN-BEM is specifically formulated for the creation of boundary element models. Second, and perhaps more important, since COMGEN was developed before PCL had undergone any significant development and prior to introduction of the X-Windows based version of MSC/PATRAN, the integration of COMGEN-BEM into the MSC/PATRAN environment and the data input for COMGEN-BEM has been significantly improved over that of COMGEN.

There are two main objectives of this paper. The first objective is to provide a detailed overview of the structure and operation of the COMGEN-BEM program. This objective is accomplished first through an overview of the boundary element methodology and COMGEN-BEM. The COMGEN-BEM overview includes the available model types, a general description of the types of input data that are required, and a discussion of how the input data is utilized by MSC/PATRAN in order to create a boundary element model. The overview is followed by a detailed description of the menus and forms utilized for data input, along with a general description of the translator which has been developed for converting the MSC/PATRAN model data into an acceptable boundary element analysis input file. The second objective of this paper is to demonstrate how MSC/PATRAN and PCL can significantly simplify the generation of boundary element models in general, and composite micromechanical models in particular.

OVERVIEW OF BOUNDARY ELEMENT METHOD

The boundary element methodology to be described here has been implemented into the NASA Lewis computer code BEST-CMS [3]. BEST-CMS includes the capability to conduct elastostatic, heat conduction and thermoelastic analyses. The major difference between the boundary element method and the more commonly used finite element method is that while in the finite element method the entire volume of a domain is discretized, in the boundary element method only the outer surface of a domain is discretized.

A composite material is modeled by discretizing the outer surface of the composite matrix using quadrilateral surface elements (eight noded elements usually working best). To model the fibers of the composite, in order to eliminate the detailed discretization which would be required in order to explicitly model the fibers of the composite, specially formulated "Fiber" elements have been developed. For these "Fiber" elements, only the centerline of the fiber is defined using line elements (three noded elements usually working best). The fiber radius is also specified in the input file. The "Fiber" elements are then placed within the outer surface of the matrix due to the BEST-CMS assumption that fiber ends cannot be free surfaces. The fiber surfaces and the variation of the field variables in the plane of the fiber cross section are represented through the use of trigonometric shape functions and closed form analytical expressions within the boundary element formulation. To calculate the variation of the field variables along the length of the fiber, numerical integration is performed. The fibers are currently assumed to have a circular cross section. However, improvements are currently being made to the BEST-CMS code in order to allow for elliptical fiber cross sections to more accurately reflect the geometry seen in woven composites. By utilizing these formulations, the mesh complexity involved in constructing a three-dimensional composite micromechanical model is significantly reduced, particularly for models with complex interior fiber architectures.

Several other aspects of the BEST-CMS computer code are important to discuss in terms of the model generation process implemented within COMGEN-BEM. In terms of geometry, the boundary element model is divided up into self-contained, closed subregions called Generic Modeling Regions (GMR). Each model must contain at least one named and defined GMR. By dividing up the model into such GMRs, the computational efficiency of the BEST-CMS code can be improved. Each GMR is given a set of material properties, and the nodal coordinate and element connectivity data is defined for each GMR. However, within each GMR definition the data for the nodes and elements which model the matrix is defined separately from the data which defines the fiber. To define the interface region where two GMRs intersect in a model, the elements of each GMR which belong to the interface are identified and defined within the input file.

One other important point to note about the boundary element formulation is the manner in which the loads and boundary conditions are defined. Force loading is applied by applying a traction load over an element or a group of elements, unlike in the finite element method where point forces are applied at nodes. Additionally, displacement or temperature boundary conditions are also specified as being applied over an element or a group of elements, instead of just being applied at a point as in the finite element method.

OVERVIEW OF COMGEN-BEM

There are several model types available within COMGEN-BEM. All of the models are three-dimensional. The composite models are based on the concept of a unit cell, the smallest region for which the response of the model is representative of the response of the composite as a whole. For the laminated composite models, the unit cell consists of a single fiber embedded and centered in a matrix. For the woven composite models, the unit cell is the smallest region which fully represents the woven nature of the fiber geometry. The model types available for the laminated composite models include a one cell square model (Fig. 1) and a four cell square model (Fig. 2). For both of the basic square models, the model width (X-direction in the figures), height (Y-direction), and thickness (Z-direction) are all assumed to be equal. The four cell square model assumes that the horizontal spacing between fiber centers is equal to the vertical spacing. For both of the square models, options are available which allow the user to vary the model thickness. Options are also available for the four cell square model which allow the user to explicitly specify the horizontal or vertical spacing between fiber centers, resulting in an rectangular model. For woven composites, a plain weave model is available (Fig. 3). For the plain weave model, the user can either select a basic model where the model width, height and thickness are all set equal and the waviness ratio (ratio of curved fiber tow to straight fiber tow) is set equal to 0.5, or a more complex model where the model height and waviness ratio can be explicitly set.

All of the user input in COMGEN-BEM is accomplished through the use of menus and forms. A session file is played which sets up a "BEST-CMS" option in the main MSC/PATRAN menu bar from which the program can be executed. Once the program is executed, a form is displayed from which the user selects the model type to be generated. After the model type is selected, another form is displayed in which the user is prompted to enter the values required in order to generate the model geometry. For all of the forms, checks are made to ensure that valid input values are given, and the user is prompted to repeat the data entry if an invalid parameter is input. Once the geometry is input, a form is displayed which prompts the user to enter the mesh density values needed to appropriately mesh the generated geometry. For selected one cell square and four cell square models the user is then given the option to rotate the fibers by 90°. Next, the material property data for both the fiber and matrix is entered through an appropriate form. Finally, a form is displayed which allows the user to input data relating to the loads and boundary conditions. The boundary element model is generated based on the input data provided on the user input forms. The data is assigned to appropriate PCL variables, and the PCL statements utilized to create a model are then executed, with the variable names replacing specific data values. Once the model is fully generated, the user can then invoke a provided translator from the "BEST-CMS" option in the main menu bar in order to create an appropriate input file which can be analyzed using BEST-CMS.

INPUT DATA AND FORMS

To execute COMGEN-BEM, after a new database has been created a session file is executed which establishes the analysis preference for BEST-CMS, as well as compiling the functions required to execute COMGEN-BEM. As is seen in Figure 4, the session file also sets up the option "BEST-CMS" in the main menu, from which a pull down menu can be accessed. To execute COMGEN-BEM, the option "Model Generator" is selected from the pull down menu.

Once COMGEN-BEM execution is initiated, the form shown in Figure 5 is displayed, from which the user can select which type of model is to be generated. For the one cell square and four cell square models, two options are available; one option in which the model thickness is automatically set equal to the model height and width, and another option in which the user can explicitly define the thickness of the model. For the rectangular models, the user can choose to either explicitly define the horizontal spacing/distance between fiber centers (Four Horiz) or the vertical spacing/distance between fiber centers (Four Vert). For both of these models, the user again has the option to select either a default model thickness or an option which allows the explicit definition of the model thickness. For the plain weave woven models, the user can select the basic model in which the model height, width and thickness are all set equal, and the waviness ratio (ratio of curved fiber to straight fiber) is set equal to 0.5. Alternatively, the user can select an option in which the ply height and tow waviness ratio is explicitly defined by the user. The user selects the desired model by clicking on the diamond next to the model type which is desired, and then clicking on the "Apply" button. For all of the COMGEN-BEM forms, selecting "Apply" is what tells the program that the user is finished entering data for the particular form, and further program execution is to begin. On this form, as in all of the forms in COMGEN-BEM, if the "Cancel" button is selected, the current form will be hidden and the program execution will be terminated. If the user does not select any of the options before the Apply button is selected, a form will be displayed (Fig. 6) which will prompt the user to select an option. If the user wishes to select an option and continue, the "Yes" button on this form should be selected. The user will then be returned to the model selection form at which time a selection can be made. If the "No" button is selected, all of the currently displayed forms will be hidden and the program execution will be terminated.

Once the model type is selected, a form is displayed which allows the user to define the parameters from which the model geometry will be generated. A sample geometry input form for the laminated composite models is given in Figure 7, and a sample geometry form for the woven composite models is given in Figure 8. For both of the model types, the numerical values of the geometric parameters are entered into the databoxes. The exact values which will need to be entered will be dependent on the model type which was selected. For all of the models, the fiber volume ratio (the ratio of the volume of the fibers to the total volume of the composite) and the fiber diameter will be requested. The other required input parameters will vary depending on the model type which is selected. The input can be in any unit of measurement desired, but the units must be kept consistent throughout the entire data input process. For all of the models, each of the geometric parameters must be greater than zero, and a form similar to that in Figure 6 will be displayed if such is not the case. In this situation, the user will be prompted to enter valid data. If the user chooses to do so (by selecting "Yes" on the error form), the user can return to the geometry input form and reenter valid parameters. If the fiber distance is specified for the laminated rectangular models or the model height and waviness ratio are specified for the woven composite models, mathematical checks will be performed in order to check that a valid model will be generated and that invalid conditions (fiber overlap, ply height being less than fiber diameter, etc.) will not be encountered. If such conditions are encountered, an error form will be displayed and the user will be prompted to reenter the geometric values.

When valid geometric data has been input, the model geometry will be generated based on the input given by the user. The geometry of the matrix outer surfaces is generated using two-dimensional patches, and the fiber geometry is generated using lines. For the laminated composite models, one line is used to model each fiber. For the woven composite models, three lines are used to model each curved fiber in order to approximate the curvature. In order to satisfy the BEST-CMS assumption that fiber ends are not free surfaces, the lines which represent the fibers are scaled and placed such that they lie entirely within the outer surface of the model. To identify the GMRs required for BEST-CMS, for the one cell square and woven models the entire model is considered to be one GMR and is defined as GMR1. For the four cell square and four cell rectangular models, the bottom layer of fibers and matrix is defined as GMR1 and the top layer of fibers and matrix is defined as GMR2 (Figure 9). It is important to again note that each GMR is a self-contained closed subregion.

After the geometry is generated, a form is displayed which allows the user to define the mesh density information required to create a boundary element mesh of the model. A sample mesh input form is shown in Figure 10. The user is required to input the number of elements desired in the horizontal and vertical directions for the patches on the front and back faces of the model (X-Y faces in Figures 1-3). The horizontal and vertical directions are defined in Figure 11 for the one cell square and woven models, and in Figure 12 for the four cell square and rectangular models. It is important to note that for the four cell models the mesh density is defined for only one of the four cells. Therefore, a mesh density of one element in the horizontal or vertical direction would result in a total of two elements across the entire width or height of the composite (one element per cell). The mesh density along the thickness direction for the patches is also defined by the user. For the lines which model the fiber, the user is prompted to enter the number of elements with which to mesh each fiber. For the woven composite models, since each fiber is composed of three lines, a minimum of three elements must be used to mesh each fiber, and the number of elements used to mesh each woven fiber must be a multiple of three. All of the mesh density inputs must be greater than zero. If invalid data is input, an error form (similar to Fig. 6) is displayed, and the user is prompted to reenter the mesh density data.

Once valid mesh density data is input, the boundary element mesh is generated. Eight noded quadrilateral elements are utilized to mesh each of the patches, and three noded line elements are used to mesh each of the lines. Since the BEST-CMS input file requires the nodes and elements which model the matrix to be defined separately from the nodes and elements which model the fibers, the node numbers of the fiber nodes are set to be greater than or equal to 5000, and the element numbers of the fiber elements are set to be greater than or equal to 1000, which allows for easy differentiation of fiber nodes/elements and matrix nodes/elements. For the four cell square and rectangular models, the fiber node numbers in the top layer (GMR2) are specified to be greater than or equal to 6000, and the fiber element numbers in the top layer are specified to be greater than or equal to 2000. This detailed numbering scheme is utilized by the MSC/PATRAN-BEST translator in writing out the input file. Once the mesh is generated, each GMR is equivalenced separately. For the four cell square and rectangular models, the separate equivalencing of each GMR is important since BEST-CMS requires that the nodes and elements which belong to each GMR are unique, have a unique number, and are not shared between GMRs.

For the laminated composite models (one cell square, four cell square, rectangular) in which the model thickness is specified by default to be equal to the width, the user is given the option to rotate the fibers by 90° in the x-z plane. A sample form from which this information can be input is shown in Figure 13. For the one cell square model, the user is only given the option as to whether or not the one fiber is to be rotated. For the four cell square and rectangular models, the user is given the option to select whether or not the fibers in the top layer (GMR2), the bottom layer (GMR1) both layers or neither layer should be rotated. The desired option is chosen by selecting the diamond next to the option wanted in the form. If no option is chosen, an error window is displayed which prompts the user to choose one of the options. Only 90° rotations are possible at this time due to the BEST-CMS assumption that fibers must have a circular cross section along their entire length. Therefore, any fiber rotation other than 90° would result in difficulties in fitting the fibers within the matrix while still maintaining a constant fiber volume fraction. Currently under development for BEST-CMS is an oblique fiber cross section formulation, which would allow more flexibility in rotating fibers.

The material property data is the next set of information which needs to be input by the user. The form that is required for input of this data is shown in Figure 14. The data is entered in the provided databoxes. For the matrix, the user is required to enter the reference temperature at which the property data was taken, the elastic modulus, the Poisson's ratio, density, thermal expansion coefficient, the thermal conductivity and the specific heat. While all of this data may not be required for every type of analysis (for example, the thermal properties are not required for a purely elastic analysis), all of the data is still requested in order to ensure that for whatever type of analysis the user wishes to perform, the required property data will be present. At this time, only the properties for one temperature level can be entered, i.e. the properties cannot be set to vary with temperature. If such a condition is required, the user would have to explicitly input that information into the input file. Similarly, only elastic, isotropic material property data can be defined. For the fiber, the user is required to enter the elastic modulus, the thermal expansion coefficient, the thermal conductivity and the fiber diameter. While the fiber diameter was entered before when the geometric data was constructed, this second entering of the fiber diameter is required so that it can be defined as an element property of the fiber elements. When the input data for the fiber elements is then written to the input file, the fiber diameter information is then read directly from the element property definition. While ideally the user should only have to input the fiber diameter data once, efforts to transfer the information from the PCL functions used to generate the geometry to the functions used to define the material and element property data, such efforts were not successful. The entered fiber diameter must be greater than zero. If such is not the case, an error form similar to that shown in Figure 6 will be displayed, and the user will be prompted to enter valid data. If the entered data for the modulus, Poisson's ratio, density, thermal expansion coefficient, thermal conductivity or specific heat is set to be less than or equal to zero, an error form similar to that shown in Figure 6 will be displayed. At this point, the user should check all of the defined data. If the material to be analyzed actually is supposed to have a negative or zero property for the material property, the user should select "Yes" on the error form to indicate that yes, the properties are correct. The given material properties will the be assigned to the model. If the user made an error in entering the data and wishes to change the input, the user should select "No" on the error form and the user will be allowed to reenter the data. Likewise, if the entered Poisson's ratio is outside of the range from -1 to 0.5, the normal range for isotropic elastic materials, an error form will be displayed, and the user will be prompted to indicate whether or not the entered Poisson's ratio data is valid.

Once the material property data is entered, the material/element properties are assigned to the model. The material properties for the matrix are assigned to the name "MAT_1", and the material properties for the fiber are assigned to the name "MAT_F". These property names are then searched for by the MSC/PATRAN-BEST translator in order to write out the appropriate data in the appropriate place in the input file. In BEST-CMS, for each GMR an element must be identified which establishes the outer normal direction and its relationship to whether the ordering of the nodes of the defined element is counterclockwise or clockwise. One matrix element in each GMR is given the property name of "100", which is utilized by the translator to identify which element establishes the positive outer normal direction for each GMR. Four the four cell square and rectangular models, the elements which lie on the interface between the bottom layer (GMR1) and the top layer (GMR2) are given a property name of "50", to identify these elements as interface elements for the translator. An element property named "60" is then given to the fiber elements. The fiber diameter is defined as part of this element property definition.

The final set of data that needs to be input is the information regarding the loads and boundary conditions that are to be applied to the model. The form that is utilized to input this data is shown in Figure 15. Each load or boundary condition that is to be applied to the model is applied one at a time. First, the user is to define whether a pressure, displacement or temperature boundary condition (the three boundary conditions currently available within COMGEN-BEM) is to be applied for the current loading condition. Pressure loadings are used instead of point force loadings since in the boundary element method tractions are applied to elements instead of point forces being applied to nodes. Next, the user selects which plane on which the current boundary condition is to be applied. Since the unit cell models are assumed to be very small compared to the overall dimensions of the structure, the boundary conditions are assumed to be applied to an entire face of the model and not to vary over that face. Referring to Figures 1-3, the Front and Back faces refer to the faces of the model parallel to the X-Y plane, the Top and Bottom faces refer to the faces parallel to the X-Z plane, and the Right and Left faces refer to the faces on the Y-Z plane (see Figure 16 for a schematic). If the user does not select a load type or a load plane, an error form similar to Figure 6 will be displayed. The user is prompted to enter the numerical value of the current load or boundary condition in the appropriate databox. For a pressure or temperature load, the magnitude of the pressure or temperature to be applied is input. A positive pressure indicates a tensile load, while a negative pressure indicates a compressive load. If inappropriate blanks or alphabetical characters are entered in this databox or the load value is left blank, an error form is displayed. For a displacement load, the displacements are to be entered in the vector format typical of MSC/PATRAN, i.e. "<x-value,y-value,z-value>". If proper vector format is not utilized, the user will be notified that an inappropriate format was utilized. The user is also prompted to enter an alphanumeric name for the loading condition. A unique name must be given for each load, but otherwise no other restrictions are placed on the load name. Finally, the user is to indicate whether or not any more loads or boundary conditions are to be applied to the model. If the user answers "Yes" to this prompt, after the load is applied the input form will still be displayed, at which time the user can enter information to input another load or boundary condition. If the user selects "No", after the current load is applied the form will be hidden and the program execution will be terminated.

Once the model has been generated, the user can then invoke a MSC/PATRAN-BEST translator which has been developed in order to allow for the writing out of an appropriate BEST-CMS input file from the MSC/PATRAN model. The translator is invoked from selecting the PAT-BEST option from the BEST-CMS pull down menu in the main menu bar (Fig. 4). Once the translator is executed, default analysis control information is written out, the material property data is defined, the nodal and element data for each GMR is written out, the GMR interface elements are specified, and the load and boundary condition data is written out. After a few minor edits the tailor the input file for the specific problem to be solved, the input file can then be submitted to BEST-CMS for analysis. Future modifications will involve making the translator more interactive, allowing the user to select such items as analysis title, analysis type and fiber/matrix interface conditions. Additionally, a BEST-MSC/PATRAN translator will be developed.

DISCUSSION

In creating the COMGEN-BEM program, several interesting aspects of the PCL language were utilized. To move from one input form to another, a "cascading" methodology is utilized where the final statements of the "apply" function of one class display the input form for the next appropriate class and hide the input form for the current class. This methodology was utilized since difficulties were encountered in trying to use global variables to regulate the display of the input forms and in transferring information between classes. By using the cascading methodology, no information needed to be transferred between classes. This methodology was also utilized in the error trapping routines. In the "apply" function for each class, if the input read from the input form is invalid, the UI_READ_LOGICAL function is utilized to allow the user to indicate if the program execution is to continue. If the response from the function is "TRUE", the input form for the current class is redisplayed. If the response is "FALSE", the input form is hidden, thus terminating the program execution.

In the geometry generation, the parameters input by the user are converted to the appropriate values needed by MSC/PATRAN (model width, model height, etc.) through the use of mathematical expressions. The PCL geometry functions (asm_const_grid_xyz, etc.) are then executed, with an appropriate variable replacing the explicit values in the commands. The correct sequence of commands were generated by creating baseline models using default values for model height, width, etc. The generated journal files were then edited to replace the explicit values in the commands with variables where appropriate. To simplify the geometry creation for the four cell square and rectangular models and to insure that the geometric entities are placed in the correct GMR (group), the geometry for the bottom layer is created and assigned to the group "GMR1". The group "GMR2" is then created and set to be current, and the geometry for "GMR2" is generated by transforming the geometry of "GMR1" upward by the amount of the layer height. In this manner, the geometry for GMR2 is created without having to duplicate the geometry creation process used for GMR1.

To mesh the model, several techniques are utilized in order to maintain unique nodes and elements in each GMR, and to explicitly set the node and element numbers of the "fiber" elements. For the one cell square and woven models, the fiber lines are meshed after the surface patches. In this manner, the node and element numbers for the fiber elements can be explicitly set in the meshing command without affecting the node and element numbers of the surface elements. For the four cell square and rectangular models, the surface patches for each GMR are meshed (separately), followed by the fiber lines (giving explicit numbers to the fiber nodes and elements). In order to maintain unique nodes and elements for each GMR, the groups corresponding to each GMR are equivalenced separately.

In rotating the fibers, in order to rotate all of the entities connected to the fibers (lines, nodes and elements) without losing any of the entity numbering, a special technique is utilized. First, the fiber entities of a GMR are moved to a new temporary group. Next, the nodal coordinates for all of the surface nodes of the GMR are examined in order to determine the length, height and width of the GMR (the maximum x, y, and z nodal coordinates found). The axis of rotation (a vertical line running through the center of the GMR) is then determined using these coordinate values. The temporary group containing the fiber entities is then rotated by 90° using a group transform command, thus rotating the fibers. The rotated fiber entities are then moved back to the GMR from which they originated, and the temporary group is deleted.

For the material properties, the data entered for the matrix and the data entered for the fibers are converted into two string arrays, MATPROPS for the matrix properties and FIBPROPS for the fiber properties. Each individual property value is one element of the appropriate array. This methodology is used in order to simplify the process of using the material data in the material.create command. By using string arrays, only the array name has to be specified in the portion of the material.create statement where the property values are specified. In order to complete the arrays for the creation of the fiber material by MSC/PATRAN, any property which is not explicitly defined (or needed) for the complete BEST-CMS material definition is given a dummy value.

A similar array technique is utilized for the application of the loads and boundary conditions. String arrays are established which contain the total information on the load value needed for the loadsbcs_create command. Depending on the load type (pressure, displacement, or temperature), the input load value is assigned to an appropriate element of the load array, with the other array elements being explicitly set to appropriate values depending on the load type. The load array is then given as the parameter in the loadsbcs_create statement to identify the load values. This methodology simplifies the actual loadsbcs_create statement significantly. The loads are applied to geometric patches instead of BEM entities since the geometric patch IDs cannot be varied by user action, while the node and element numbers will vary based on the user defined mesh density.

Different loadsbcs_create commands are utilized depending on the load type and the load plane, in which all of the data except for the actual load value is explicitly set depending on the load type and load plane selected. In order to repeat the display of the input form for the entry of multiple loads, the user entry of whether or not the load entry is complete is used as a flag. If this flag indicates that load entry is not complete, the input form for load entry is redisplayed, otherwise the form is hidden.

CONCLUSIONS

COMGEN-BEM was designed to allow a user to quickly generate a boundary element model of a composite material at the micromechanical level with a minimum of user input required. Additionally, the user does not need to have a detailed knowledge of either MSC/PATRAN or BEST-CMS in order to generate an acceptable model. The flow of the program, in which the user is sequentially moved from geometry generation, to mesh definition, to the specification of material properties to the application of loads and boundary conditions, provides a very systematic method to generate a model with a reduced possibility of error. With the program being written in PCL and executing directly within the MSC/PATRAN environment, the user is able to operate in a familiar environment of menus and forms. Additionally, using menus and forms to enter the data allows the user to check and adjust their input before submitting the data to the program. The error checks provided within COMGEN-BEM further reduce the possibility of invalid data being entered. The generated models are automatically in the correct form to be translated into a valid BEST-CMS input file, and the input file generation can be quickly accomplished using the provided translator.

When a parametric study is to be performed, such as studying the effect of varying the fiber volume fraction on the response of a material, utilizing an automatic model generator can be much more efficient than having to create each model from scratch. Additionally, since many users are unfamiliar with the boundary element method and boundary element modeling, an inexperienced or infrequent user could very easily make an error in the model creation process. Besides the general unfamiliarity with boundary element modeling, users could easily encounter difficulties in incorporating some of the unique parameters of BEST-CMS (GMRs and their interfaces, normal element specification, etc) into their model. By utilizing COMGEN-BEM, incorporating these parameters is automatically taken care of.

In addition to demonstrating the ability of PCL and MSC/PATRAN to simplify the generation of boundary element models of composites, some more general benefits of utilizing MSC/PATRAN are demonstrated by this program. The program could easily be modified to allow for the automatic generation of boundary element models of other relatively simple geometries such as an airfoil or fan blade, or pressure vessel. In these cases, the geometric parameters to be input by the user would have to be changed to parameters such as thickness and camber if an airfoil geometry is to be generated, but such changes would be relatively simple. Additionally, the mesh density, material property and load/boundary condition input would have to appropriately modified in order to have the user input reasonable parameters. The most difficult part of this process would be in creating the basic model from which the session or journal file would have to be transformed to a parametric form to allow for setting the appropriate geometric or other parameters to variables. Finally, the program could even be modified to create finite element models of a simple geometry based on basic user input parameters. The important point to be gained from this effort is that a process has been established to allow a user to create a complete boundary element (or finite element) model of a simple geometry, including everything from geometry to load definition, based solely on the entry of a few simple parameters by the user, and without requiring a detailed knowledge of the model generation process.

ACKNOWLEDGMENTS

The second author gratefully acknowledges the Structural Mechanics Branch of NASA Lewis Research Center and the staff of the NASA/OAI Collaborative Aerospace Internship and Fellowship Program for providing the opportunity and the funding to conduct the work presented in the paper.

REFERENCES

[1] MSC/PATRAN User's Manual, Version 1.3, PDA Engineering, Costa Mesa, CA, October, 1994.

[2] Melis, M.E., "COMGEN: A Computer Program for Generating Finite Element Models of Composite Materials at the Micro Level," NASA Technical Memorandum 102556, NASA Lewis, Cleveland, OH, 1990.

[3] Henry, D.P., Banerjee, P.K., and Dargush, G.F., "Development of Boundary Element Methods for Ceramic Composites," NASA Contractor Report 195326, NASA Lewis, Cleveland, OH, 1994.

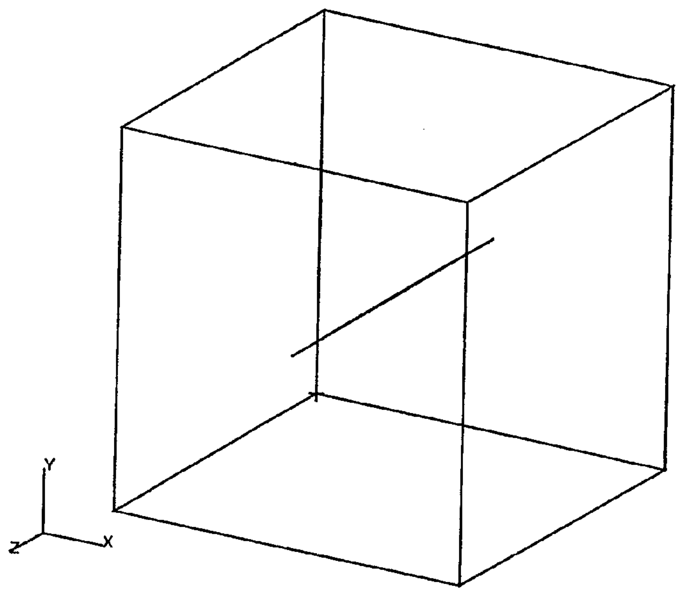

Fig. 1: One Cell Square Model

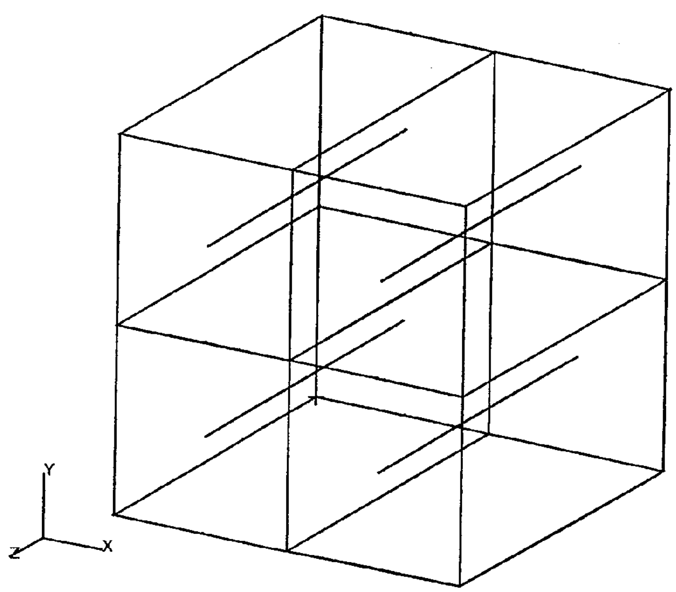

Fig. 2: Four Cell Square Model

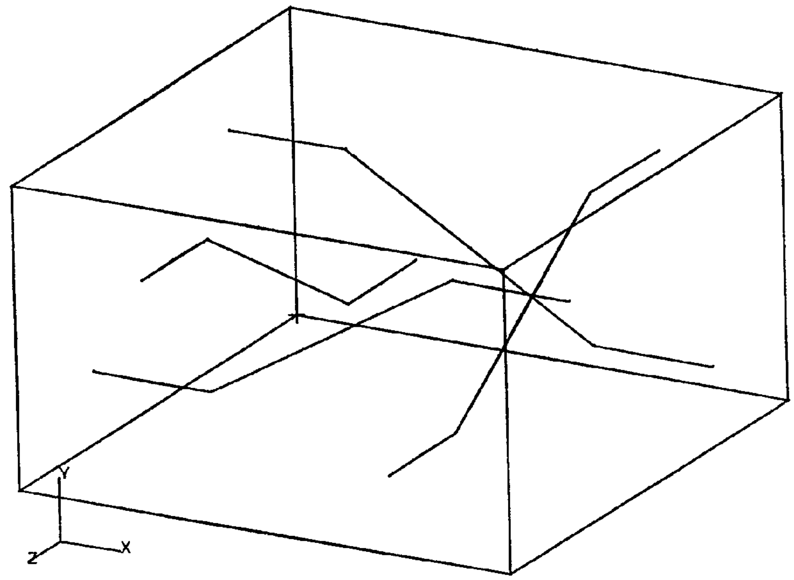

Fig. 3: Plain Weave Woven Model

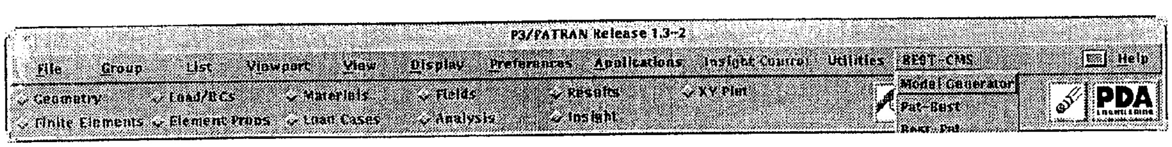

Fig. 4: Main Menu Bar with BEST-CMS Option

Fig. 5: Model Type Selection Form

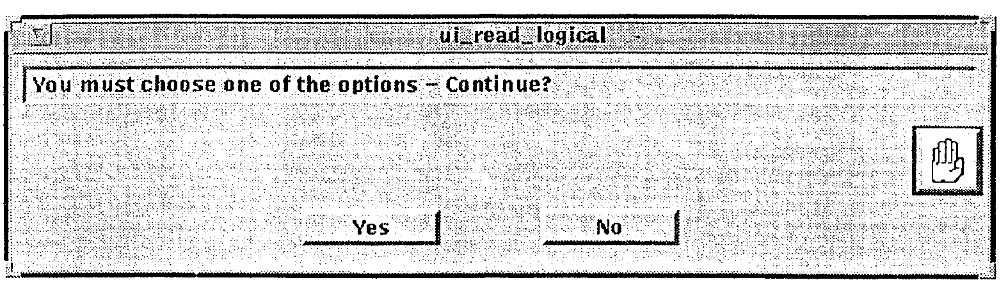

Fig. 6: Model Type Selection Error Form

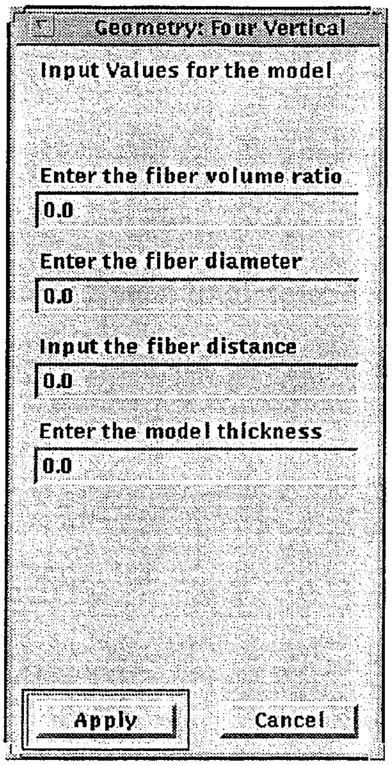

Fig. 7: Geometry Input Form for Laminated Composites

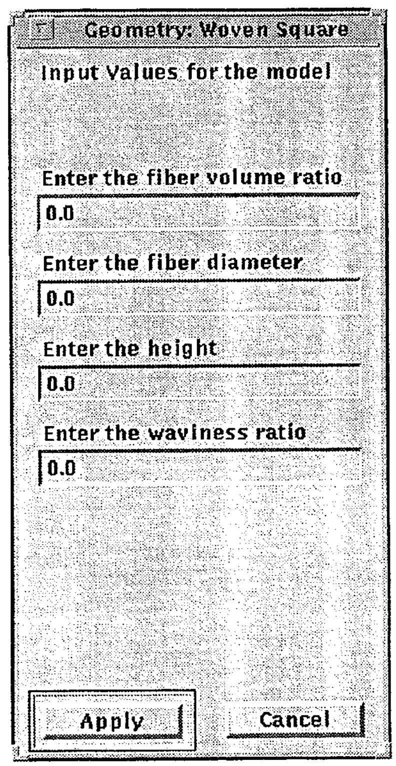

Fig. 8: Geometry Input Form for Woven Composites

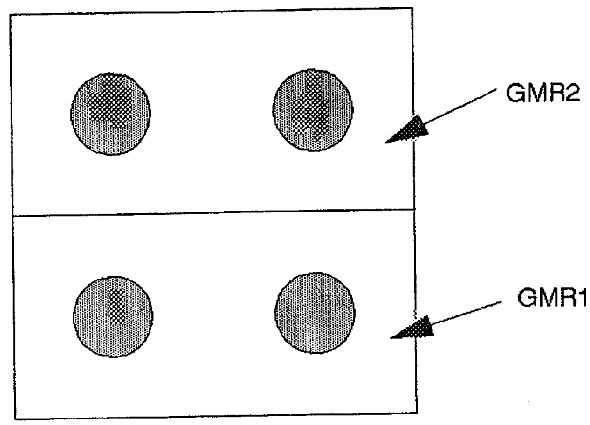

Fig. 9: Generic Modeling Region Definitions

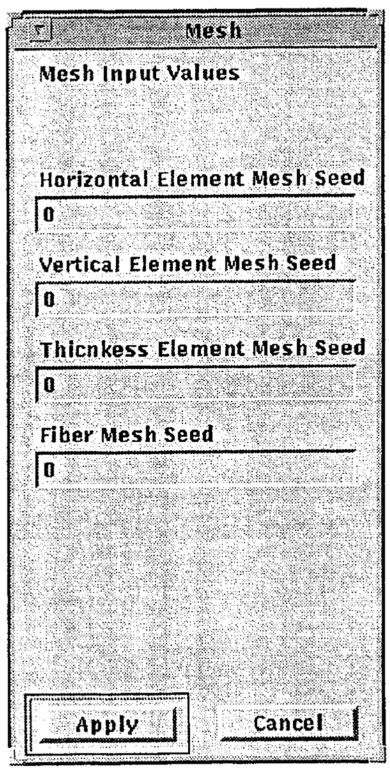

Fig. 10: Mesh Density Input Form

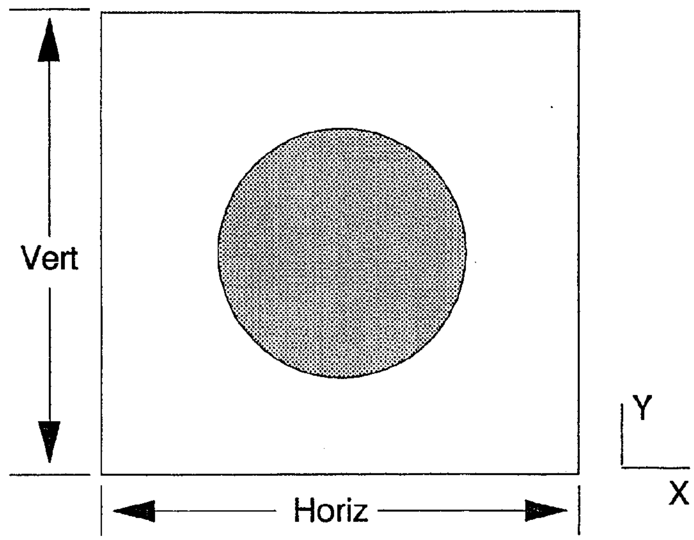

Fig. 11: Mesh Density Directions for One Cell Square and Woven Models

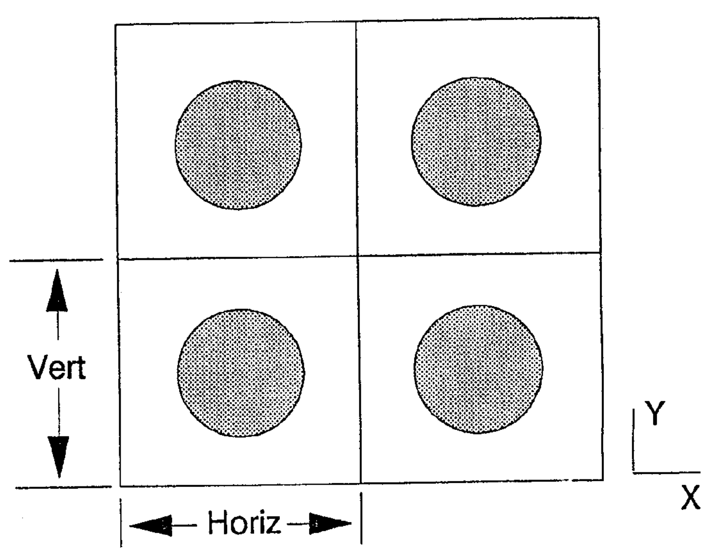

Fig. 12: Mesh Density Directions for Four Cell Square and Rectangular Models

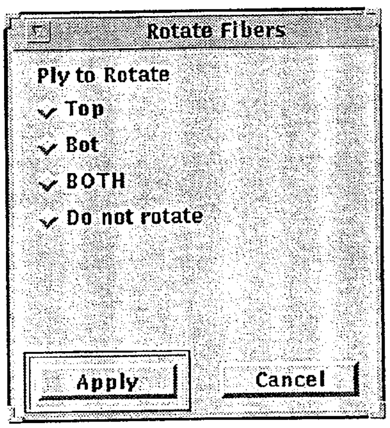

Fig. 13: Fiber Rotation Input Form

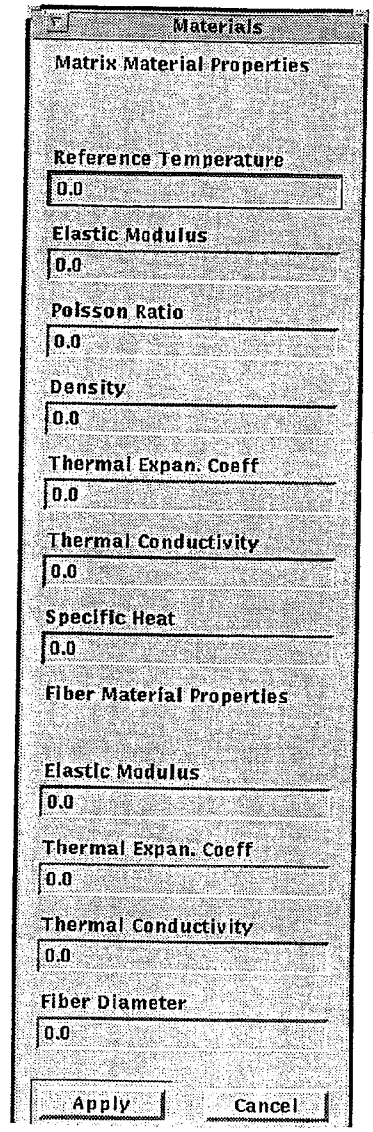

Fig. 14: Material Property Input Form

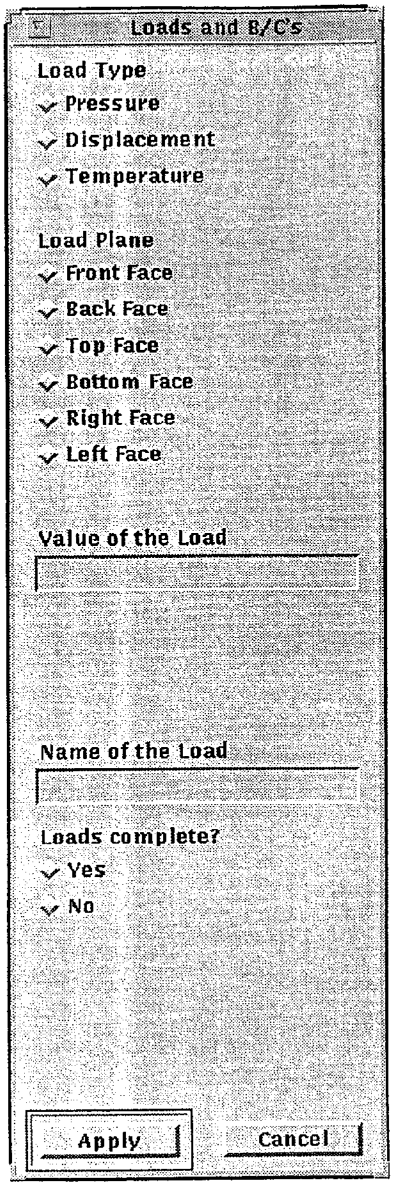

Fig. 15: Load/Boundary Condition Input Form

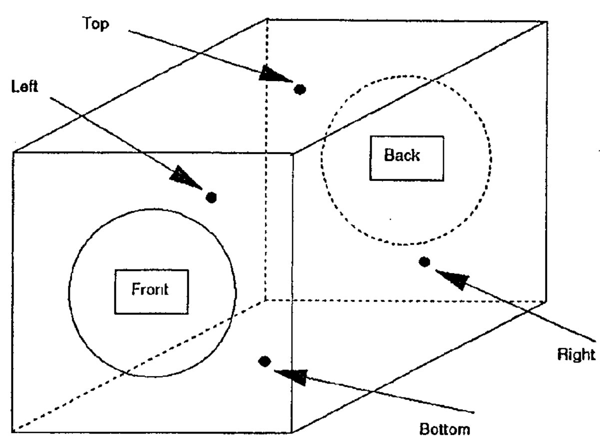

Fig. 16: Schematic of Load Planes