ERDC/GSL TR-03-17

US Army Corps of Engineers®
Engineer Research and Development Center

Research, Development, Testing, and Evaluation Program

Pavement Model Generation Interface

Report 1 – Formulation and Operation of the System

Barton W. McPheeters and Donald M. Smith

September 2003

Geotechnical and Structures Laboratory

20031104 022

Research, Development, Testing, and Evaluation Program

ERDC/GSL TR-03-17
September 2003

Pavement Model Generation Interface

Report 1
Formulation and Operation of the System

Barton W. McPheeters

MSC Software Corporation
2 MacArthur Place
Santa Ana, CA 92707

Donald M. Smith

Geotechnical and Structures Laboratory
U.S. Army Engineer Research and Development Center
3909 Halls Ferry Road
Vicksburg, MS 39180-6199

Report 1 of a Series

Prepared for U.S. Army Corps of Engineers
Washington, DC 20314-1000

ABSTRACT:

Today's pavement design calls for more emphasis on pavement performance prediction. This is a far more complex task than traditional designs that strive to provide safe thickness and specifications for the material. Modern computational mechanics provide application tools to deal with this new challenge, provided material models are created that predict cumulative deformations under traffic loading. The most theoretically rigorous analytical capabilities available today are found implemented within many of the commercial general-purpose finite element (FEM) computer programs. In order to correctly apply these analytical methods, it is necessary to produce FEM models of pavements systems that can be easily changed to allow for parameteric studies of the effects of load, environment, layer thickness, material properties, geometric properties, and boundary conditions. The Pavement Model Generation Interface (PMGI) is a unique tool created expressly for that purpose. It allows pavement analysts to crate FEM grids and load steps for pavement systems with a minimal amount of effort when compared with the tedious procedures required to produce these massive 3-D grids by traditional preprocessing methods. This report documents the Formulation, Development, and Operation of the specific tools at hand, i.e., MSC/PATRAN and ABAQUS (A General Purpose Finite Element Code), in the development of an improved analytical model for pavements. This report will be the first of a series of reports documenting the PMGI. The application and examples of using the PMGI will be detailed in subsequent reports.

Contents

List of Figures

Preface

The research reported herein was sponsored by the U.S. Army Corps of Engineers through the Research, Development, Testing, and Evaluation Program, Pavements Technology Work Package, AT40. This research was conducted by the U.S. Army Engineer Research and Development Center (ERDC), Geotechnical and Structures Laboratory (GSL), Engineer Systems and Materials Division (ESMD), Airfields and Pavements Branch (APB), Vicksburg, MS.

The study was conducted under the general supervision of Dr. David W. Pittman, Acting Director, GSL, Dr. Michael J. O'Connor, former Director, GSL, Dr. Albert J. Bush, III, Chief, ESMD, and Mr. Don R. Alexander, Chief, APB. This report will be the first of a series of reports documenting the Pavement Model Generation Interface (PMGI). The purpose of this report is to document the background, theory, and operation of the PMGI. The application and examples of using the PMGI will be detailed in subsequent reports. Dr. Donald M. Smith, APB, was the project principal investigator. Dr. Smith and Mr. Barton McPheeters, MSC-Software Corporation, wrote this report.

COL James R. Rowan, EN, was Commander and Executive Director of ERDC, and Dr. James R. Houston was Director.

1 Introduction

Background

Airfield pavement design is a complex blend of relatively simple linear elastic theory, fatigue concepts, empirical relationships derived from small and full-scale tests, and pragmatic adjustments to reflect observations of in-service pavements. This philosophy served the design community well for many years as it allowed total thickness, asphalt concrete pavement thickness, and material requirements for constituent layers in the pavement to be determined to avoid a preselected level of distress in the pavement. For airfields, this level of distress at "design" failure was selected to be 1 in. of shear rutting in the subgrade or fatigue cracking of the asphalt concrete.

However, today's designers are being asked to predict pavement performance. This is a far more complex task than simply providing safe thickness and specifications for the material. To deal with this new challenge, the design community must have material models that predict cumulative deformations under repetitive aircraft loads. With heavy loading, such as may be encountered with many airfields, the nonlinear response of base course materials must be considered when predicting pavement performance. The advances made in computational mechanics have created new tools of application for this type of problem.

The rationale and impetus for development of a detailed analytical model rests in the limitations of current pavement design and evaluation procedures. Current design procedures are very much empirical and are based on a limited number of tests conducted decades ago. Current and future wheel loadings and configurations and potential pavement systems differ considerably from the empirical basis in many cases and thus may not be easily handled by these procedures. Further, the most common current analytical tools based on the Layered Elastic Method are limited by the assumptions of this method which include (a) linear elastic material, (b) circular, static, constant pressure wheel loads, and (c) fully bonded interfaces between pavement layers. A more detailed analytical model is required to overcome these limitations.

The most theoretically rigorous analytical capabilities available today are found implemented within many of the commercial general-purpose finite element (FEM) computer programs. In order to correctly apply these analytical methods, it is necessary to produce FEM models of pavements systems that can be easily changed to allow for parametric studies of the effects of load, environment, layer thickness, material properties, geometric properties, and boundary conditions. The Pavement Model Generation Interface (PMGI) is a unique tool created expressly for that purpose. It allows pavement analysts to create FEM grids and load steps for pavement systems with a minimal amount of effort when compared with the tedious procedures required to produce these massive three-dimensional (3-D) grids by traditional preprocessing methods. This study demonstrates the use of specific tools at hand, i.e., MSC/PATRAN and ABAQUS (A General-Purpose Finite Element Code),[1,2] in the development of an improved analytical model for pavements.

Scope

This document tells what the PMGI is, what you need to know to use it, and how to use it once you have everything together. The Pavement Model Generation Interface has been developed in MSC/PATRAN's customization language (PCL (PATRAN Command Language)). PMGI runs within MSC/PATRAN and provides the user with an intuitive form-driven interface to automatically build runway pavement models. The user specifies a number of parameters on the forms, and the model is built and loaded automatically.

In its most basic form, the Interface consists of a number of PCL routines delivered as source code in the .pcl files and as a compiled library in the .plb file. In addition, there are a few other files included, such as the user_messages.database file that contains the text and identification numbers of the error messages that may appear from time to time while using the modeler. What the user will see is the Graphical User Interface (GUI) that is driven by these routines. Throughout the manual, the words aircraft and plane will be used interchangeably.

The user must first open an MSC/PATRAN database and then invoke the Pavement Modeler. When this is done, a small form will appear in the MSC/PATRAN window that provides options for modeling the pavement. When all of the form options have been set, the user selects the "Apply" button, and the model is generated.

Overview of the Pavement Model Generation Interface

The Pavement modeler interface executes via the MSC/PATRAN GUI from the main toolbar. Selecting the PMGI choice from the toolbar will start up all of the necessary programs and set up all of the required links.

---

[1] MSC Software, Patran Release Guide. (2000). MSC Software Corporation, Santa Ana, CA.
[2] HKS, ABAQUS User's Manual, Version 5.8. (1999). Hibbitt, Karlsa and Sorrensen, Inc., Pawtucket, RI.

PMGI assists in building the model. It does not run the model, nor does it set up analysis-specific items, such as the loadcases and loading steps. Additionally, while the model and all of the geometry, loads, and boundary conditions are stored in the PATRAN database, the PMGI parameters used to create it are not. Thus, if the user desires to create a record of those parameters, it is necessary to save the PMGI job control file manually.

On the other hand, aircraft wheel configurations and user-defined materials are automatically saved in external ASCII files. These files are independent of the PMGI session or job control files and can be copied and used elsewhere once created. They may also be edited to modify or correct them in preparation for another execution of PMGI.

2 Installation

Overview

Installation of the PMGI is pretty straightforward, although there are a few options that can make it a little more complicated. The basic steps involve extracting the files from the tar file, customizing the PATRAN p3epilog.pcl file, and setting up the menu description file.

Simple Install

This will install PMGI for all users who copy the requisite p3epilog.pcl file to their home or current directory:

a. Extract the .tar file. This will create a subdirectory called "runway" beneath the current directory. The location of this directory is unimportant. It is only necessary that you know where it is. Extract using:

```
tar xvf runway.tar
```

b. Copy the p3_user.pcl file in the new "runway" directory to your current directory where you want to run the modeler.

c. Copy the p3_user_menu.pmgi.def file to the same directory.

d. Edit this file, and change the line that looks like

```
pmgi_path = "/disk2/bmcpheet/_runway"
```

to reflect the location of your "runway" directory

e. Make sure the MSC/PATRAN utilities programs are installed and that the p3epilog.pcl file has been copied to the MSC/PATRAN home directory (.../patran75).

f. Edit the p3epilog.pcl file, and add the following line at the end of the file:

```
!!INPUT p3_user.pcl NOERROR
```

Start up MSC/PATRAN. When it starts up, there will be two extra menus on the top-level menu bar. One is "Utilities" which details the MSC/PATRAN utility programs delivered on the MSC/PATRAN CD. The other will be called "PMGI."

3 Using the Pavement Model Generation Interface

Main Form

When PMGI is started, the main form appears. The entire program is controlled from this form. The main form includes seven primary buttons (Figure 1). The buttons are arranged to follow the model building process from top to bottom, more or less. In order, the user selects or defines a landing gear wheel configuration ("Gear Type…"). Then the configuration is located on a section of pavement ("Gear Location…"). Next, the pavement is described, and any boundary conditions are applied (Pavement Dimensions…" and Boundary Options"). Next, the meshing details are specified ("Meshing Options…). Finally, the model is built ("Apply"). Each of these buttons and the forms they bring up will be described in Chapter 4. The only other button, the "Job Control Options..." button, allows the user to save a record of the model created, or to read a saved model into the program. That button will be described in this chapter.

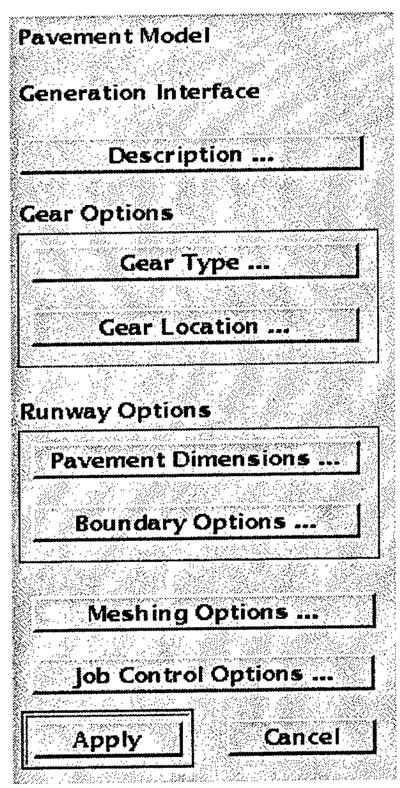


Figure 1. PMGI main form

In summary, the main form buttons do the following:

"Gear Type…" button

This form brings up the aircraft type form. This form lets the user choose which of the defined wheel configurations (which are defined by ".gear" files in the current directory (Appendix A)) will be used for the model. The form also has a button to create a new wheel configuration file.

"Gear Location…" button

As the name implies, this form lets the user locate where on the pavement each of the chosen wheel configurations will be located. Note that the location spreadsheet will be updated when new wheel configurations are chosen on the previous form. It is also necessary to have chosen at least one configuration in order to enter data on this form. Additionally, the user can select a rolling load in place of the static load. In this case, only a single plane can be specified. If more than one is chosen, only the first is used.

"Pavement Dimensions..." button

This form describes the general layout of the pavement. In particular, the length, width, and number of layers are defined. There is a further button on the form to enter data on each layer of the model. These data include the thickness, number of elements, material, and contact parameters. On the layer subform, the material choices will reflect material defined in the MSC/PATRAN database and any user-defined materials included in ".umat" files (Appendix B). There is a button on this form to create new user-defined material.

"Boundary Conditions..." button

This form lets the user choose from three different boundary conditions, including fixed boundaries, sprung boundaries (using the ABAQUS *FOUNDATION card), or infinite element boundaries. Currently, only the fixed and sprung boundaries are implemented.

"Meshing Options..." button

The options available under this menu include specification of the number of elements under the wheels, the size of elements near the wheels, and the size of elements away from the wheels. Additionally, different meshing schemes may be implemented from this menu in the future.

"Job Control Options..." button

This button brings up a menu that allows you to save your current list of parameters to a file, or to read a saved list and load all of the forms. This is useful to build a new model similar to an older one without resetting all of the form parameters, or to build a duplicate of an existing model.

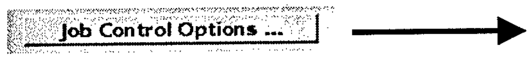
"Apply" button

The apply button actually builds the model from the parameters you have chosen and warns you about any conflicting choices you may have made. This does not automatically save your plane configuration. That must be done manually by using the "Job Control" form.

"Cancel" button

The cancel button closes the form with no further action necessary.

"Job Control" Form



The Job control form (Figure 2) allows the user to save all of the settings used to create a model, and similarly, to read a saved file into the modeler without having to type in all of the data manually. Note that neither of these operations is performed by default. This form must be explicitly called and the model saved to create a job control file. The format and data in this file are described in detail in Appendix C.

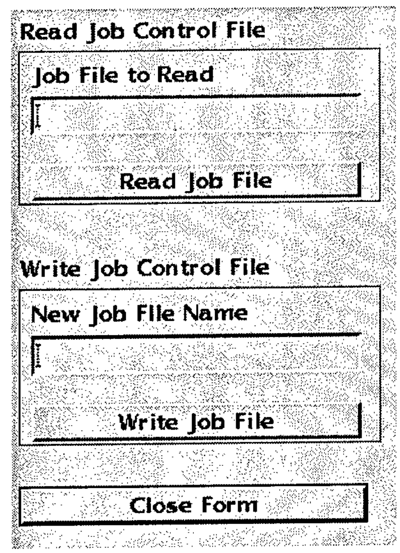


Figure 2. Job control form

Read Job Control file

"Job File to Read" databox

Enter the name of the job control file here. If a path is not specified, the program will look only in the current directory.

"Read Job File" button

Pressing this button will cause the program to look for the file named in the "Job File to Read" databox and read it. If the file does not exist, a form will appear informing the user of that. If the file exists, the file will be read and will set all of the values in the program to those specified in the file, including plane types, location, boundaries, meshing parameters, etc. If the file is incomplete, the values not included will not be set. Similarly, if data are incorrect or the wrong type, the incorrect data will simply be ignored. If you are generating a series of similar models, reading a job control file is a good place to start, as only the changes then need to be entered. If the file is empty, PMGI may lock up. If there is not a ".gear" file that corresponds to the aircraft specified in the job control file, PMGI will create one. In the case of gear configurations, the file will be complete, containing the title and all of the required data. The file name, however, will be made up to be aircraft_xx.gear, where "xx" will be the next highest numbered ".gear" file of the same name. Unknown materials will be created with no properties by PATRAN.

Write Job Control file

"New Job File Name" databox

This is the name of the new job control file to write. If a path is not specified, the file will be written to the current directory.

“Write Job File” button

Pressing this button will cause the program to search for the specified file. If it exists, the user will be prompted to overwrite the existing file. Once the preliminaries are taken care of, the file will be written. The format of this file is described in Appendix C – Job Control Files.

“Close Form” button

This simply closes the form with no further action necessary.

4 Building a Model with PMGI

Defining a Gear Configuration

A gear configuration is defined by specifying the following:

a. A name for the configuration.

b. The number of wheels that make up the configuration.

c. The x-y dimensions of the individual wheels.

d. The pressure uniformly applied over each wheel pattern.

e. The locations of each wheel relative to each other.

The **name** for the configuration can be any string of up to 40 characters. Note that only the first 20 or so will display in the listboxes in PMGI without scrolling to each side. The name should describe the configuration uniquely. For instance:

a. C-130 – STBD Gear – 0.5 MPa (74 psi).

b. Concorde - Single tire – 0.7 MPa (106 psi).

c. B-2 – Stbd Gear, aircraft #2.

A wheel is defined as a box, having an X dimension and a Y dimension. X is in the direction that the aircraft is traveling, Y is perpendicular to it (Figure 3):

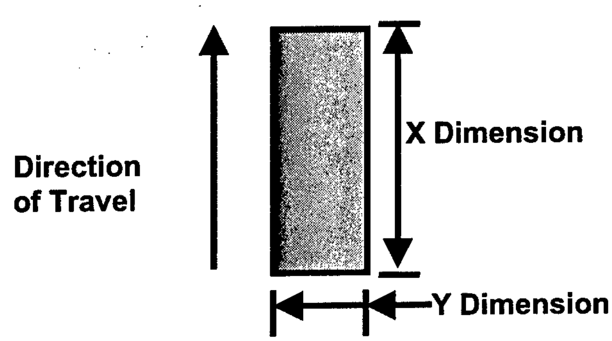


Figure 3. Wheel spacing x and y dimensions

The location of the wheels relative to each other is described with respect to a reference point. Each wheel is then located at the distance from the reference point to aft starboard corner of the wheel box (Figure 4):

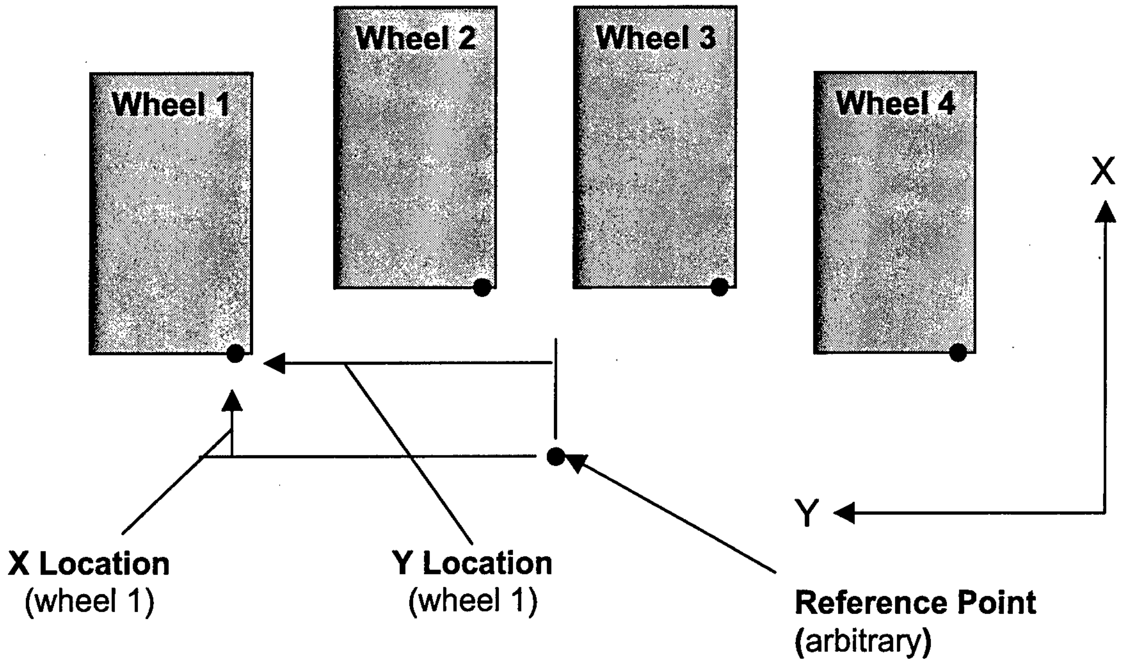


Figure 4. Location of individual wheel with reference point

This wheel description is then saved in a ".gear" file which contains all of these data in a "keyword" format that makes reading and writing the files simple. The file is arranged in eight character fields (with the exception of the title). A sample is shown in Figure 5:

```
NAME    C5-A - Single Gear
WHEELS  6
WHEELXY 18.87   15.1
PRESS   115.
COORD   1       0.      -34.00
COORD   2       0.      0.00
COORD   3       0.      53.00
COORD   4       0.      87.00
COORD   5       65.     2.50
COORD   6       65.     50.50
#2345678123456781234567812345678
```

Figure 5. Sample gear file

The meaning and format of each keyword is as follows:

NAME data
 data = configuration name (40 char)
WHEELS data
 data = number of wheels (integer)

WHEELXY data1 data2
data1 = x length of the wheels (real)
data2 = y length of the wheels (real)
PRESS data
data = uniform pressure for all wheels in the pattern (real)
COORD data1 data2 data3
data1 = wheel number (integer)
data2 = x coordinate of aft stbd corner (real)
data3 = y coordinate of aft stbd corner (real)

Data are in eight character fields, except for the name. At the moment there is minimal error checking, so wheels must be numbered 1-num_wheels, although the cards do not have to be in order.

Any field that does not have a recognized keyword is ignored.

Data are checked for the following circumstances:

a. There are numwheels number of COORD cards.

b. There are a complete set of cards.

c. The coords combined with the wheel sizes are compatible.

Once the ".gear" file is written, PMGI will automatically pick it up and list it on the "Gear Type" form. The following text describes the forms used to define and choose a gear configuration.

Appendix A contains a detailed description of the file data and formats.

"Gear Type..." Options Form

Each time you open this form, the program will scan your current directory for any files with the extension ".gear." It will then attempt to read through each file to determine if it is a valid gear configuration file. It will do this by looking for a NAME card. For each valid file it finds, it will display the value of the NAME parameter in the box at the top of the form (Figure 6).

The boxes and buttons have the following function:

"Available Gear Types" listbox
When you select a name from the top box, it will load it into the bottom box. The bottom box lists all currently selected configurations. Picking a selected name in the top box will remove it from the bottom box. Note that it is not possible to select the same name multiple times. To model a full plane of gear patterns (four identical patterns at different locations, for instance) it is necessary to make four copies of the ".gear" file in the directory. It is also worth changing the NAME field of each, since there is no other way to tell them apart within the

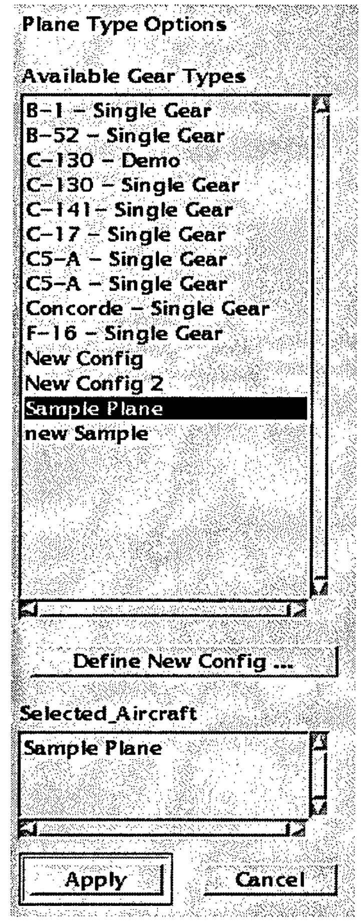


Figure 6. Aircraft options

program. Identical NAME fields may have unpredictable effects, as the configurations are sorted internally to the program via the NAME field.

"Selected Aircraft" listbox

This box is just a list of items that have been selected from the Available Gear Types listbox. Selecting aircraft names in this box will have no effect.

"Define New Config..." button

This will bring up a form that has a list of all the data needed to create a new ".gear" file. It can also be used to view or modify an existing configuration.

"Apply" button

This will save the current widget settings and store the selected aircraft list for reference by the forms that depend on this one.

"Cancel" button

This will exit the form without saving the settings. It will be as though the form had never been opened.

"New Aircraft Configuration" Form

The buttons and boxes (Figure 7) on this form have the following meanings:

"Configuration Name" databox

Enter the name that will appear on the NAME card in the ".gear" file, and in the boxes on this form and other forms that list gear configuration names. Names are limited to 40 characters, although more than about 25 characters will not appear in the data and listboxes on this and other forms.

"Configuration File" databox

This is the name of the ".gear" file that will be created when the Apply button is pushed. The suffix ".gear" will be automatically appended to the name entered in this box. Other than that, the file name is limited to 80 characters. If a full path is specified, the file will be written to that directory but will not show up in the listing, as the listing box only scans the current directory.

"Number of Wheels" option menu

Toggle this to the number of wheels that will be defined for this configuration. The program has been set up to accept a max of 10 wheels per

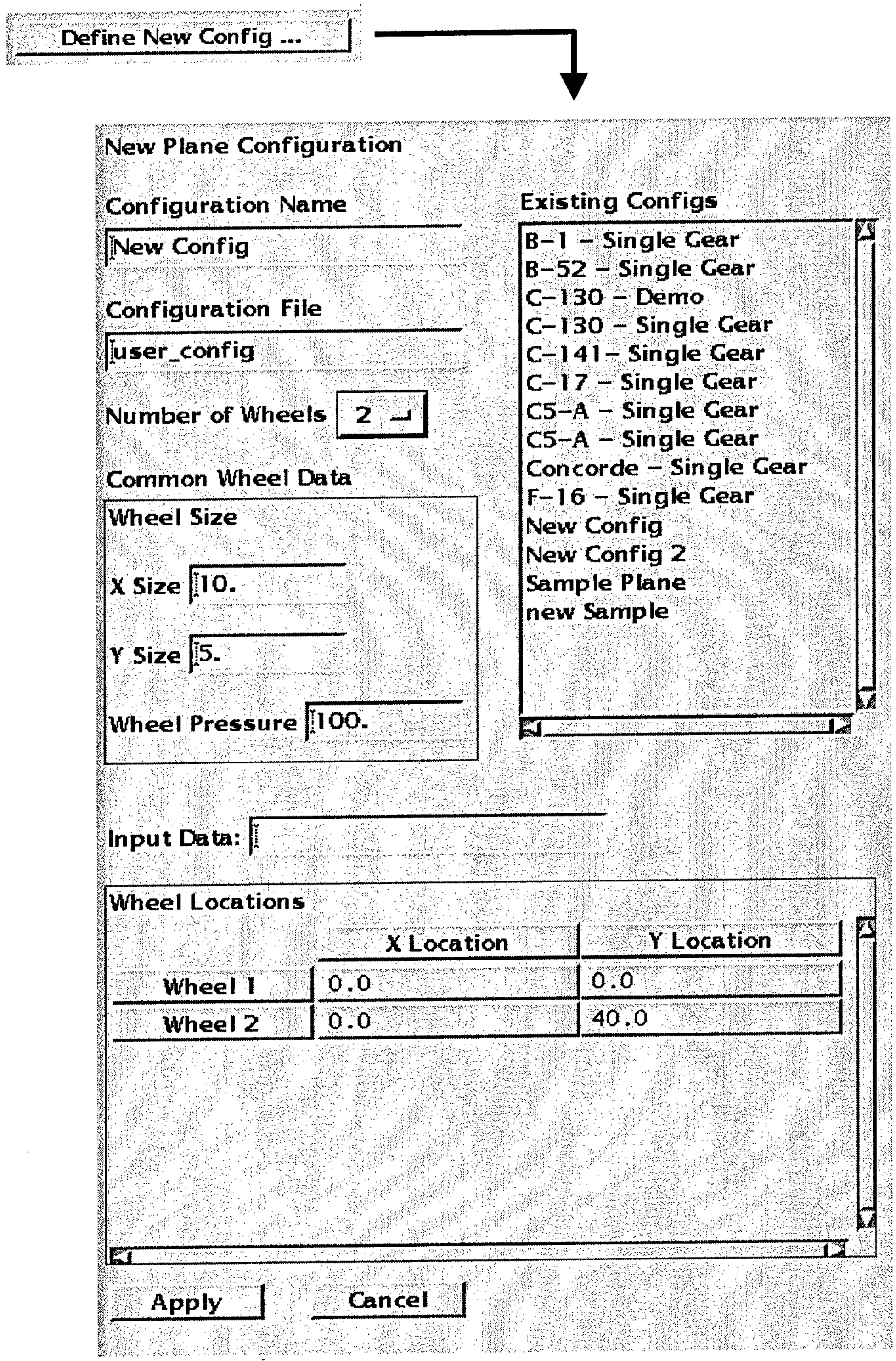


Figure 7. New aircraft configuration

configuration. If your configuration has more wheels, you can create a second configuration that has the remaining wheels in it. This would only adversely affect you if you were running a rolling analysis, where only one configuration could be selected.

Common Wheel Data

"X Size" databox

This is the X dimension of the wheel in consistent model units. X is in the direction in which the aircraft is facing.

"Y Size" databox

Y is perpendicular to the course of the aircraft.

"Wheel Pressure" databox

This is the pressure that will be applied uniformly over the wheel footprint. The pressure value should be in consistent units (e.g., If the dimensions are in inches, the pressure should be in psi.).

"Wheel Locations" spreadsheet

These will be distances from some reference point. When the pattern is placed on the pavement, the position of the reference point is what is being specified. Therefore, it is important to remember what you are using for the reference location. It may be convenient to always use a common reference point, like the center of the pattern, or at the far side of the pattern.

"Apply" button

This will write out the ".gear" file with all of the associated data and update the "existing Configs" listbox on this form and the Gear Type form.

"Cancel" button

This will close the form without making any changes.

Note that if an aircraft configuration is selected from the list, the data for it will be loaded into the data boxes. This is a good way to review or change an existing configuration, as well as to load the data for a similar configuration.

Locating a Configuration on the Pavement for Static Loads

The origin of a PMGI model is at the aft, starboard corner of the pavement. A gear configuration is located on the pavement by specifying the distance from this origin to the gear configuration reference point (Figure 8).

These dimensions are referenced on the forms as simply an X location and a Y location. Multiple aircrafts/configurations may be used for static loadings.

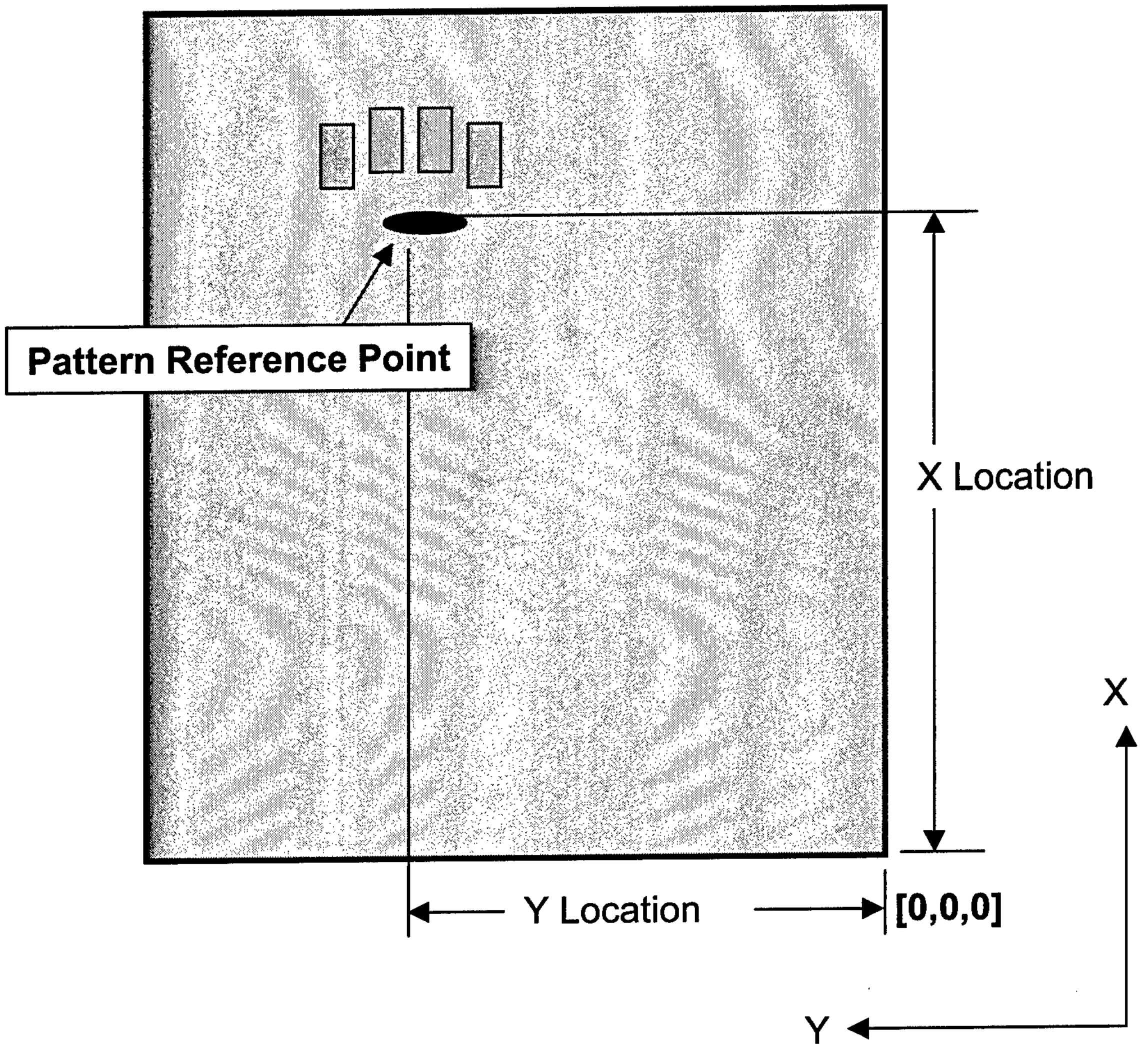


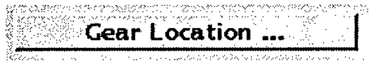
Figure 8. Plan view of load pattern

PMGI will check to assure that the aircrafts will not overlap as defined and will issue a warning message if an overlap is found.

"Gear Location…" Options Form (static load)

"Gear Location Options" switch

This switch (Figure 9) will toggle the form between the static and rolling load configurations.



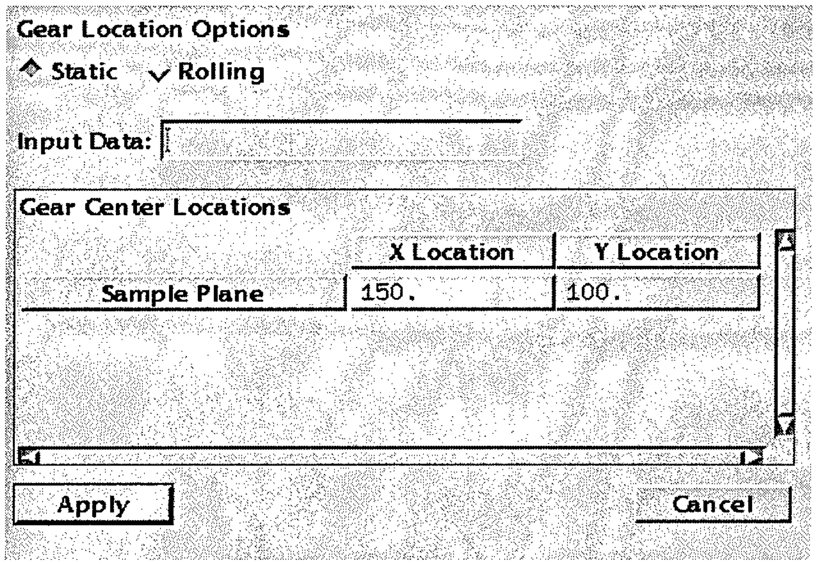


Figure 9. Gear location options switch

"Input Data" Databox and "Gear Center Locations" Spreadsheet

This form is quite simple. It has only a single databox and a spreadsheet form on which to enter these data. To use it, select a cell on the spreadsheet, then type the new value into the databox. Pressing <RETURN> will load the value into the selected spreadsheet cell and move the selected cell either down or to the right. When an occupied cell is selected, data in the cell are loaded into the databox and highlighted. This way, the data in the cell can replace without triple clicking or backspacing.

The spreadsheet will have one row for each selected configuration. If you add a gear configuration, the spreadsheet will update when you close the form and reopen it.

"Apply" button

This will save the values that have been typed into the spreadsheet. If a configuration is added after "Applying" this form, data that had been previously "Applied" will be preserved where possible on the newly displayed form.

"Cancel button"

This will close the form without saving any of the data entered.

Locating for Rolling Loads

A gear configuration is located for rolling loads similar to the method for static loads. The difference is that a starting point and an ending point are both specified. Note that only one plane may be used for a rolling load (Figure 10).

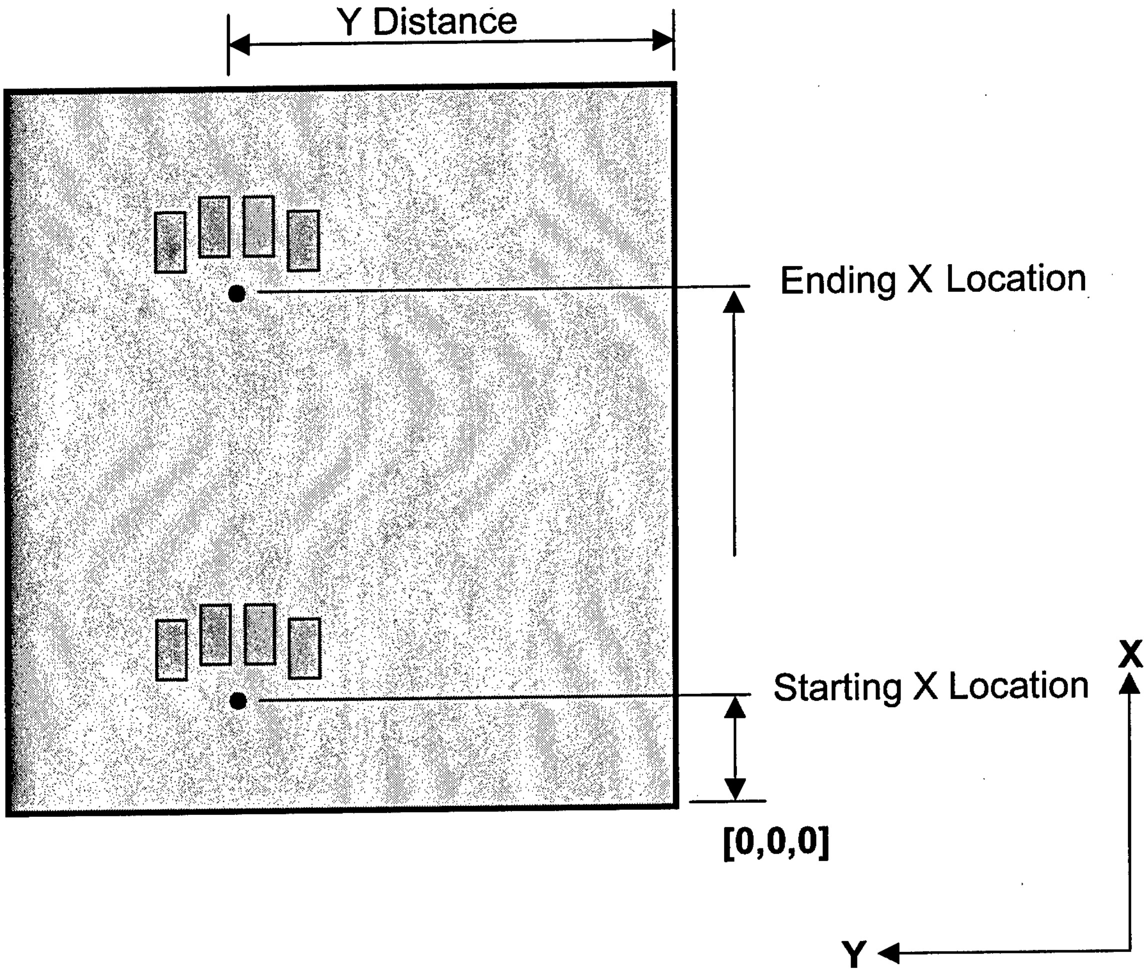


Figure 10. Gear configuration

"Gear Location…" Options Form (rolling load, Figure 11)

"Y Distance" databox

This is the location of the gear pattern on the pavement in the direction perpendicular to the aircraft's direction of travel.

“Starting X” databox
“Ending X” databox

These are the starting and ending locations of the gear pattern as it rolls down the pavement.

“Wheel Thickness” databox

This is the thickness of the shell elements used to model the wheel “slider.” In general, thinner elements converge slower but more uniformly apply the pressure. Elements that are too thin (.001 for example) tend to be unstable. Elements that are too thick may apply the pressure load around the edges on the wheel, rather than uniformly across it.

“Sliding Mu” databox

The contact surface that describes the interaction of the wheel slider and the pavement surface requires a coefficient of friction to be entered. Because we are loading the surface with a fixed displacement, the effect it has on the model is unknown.

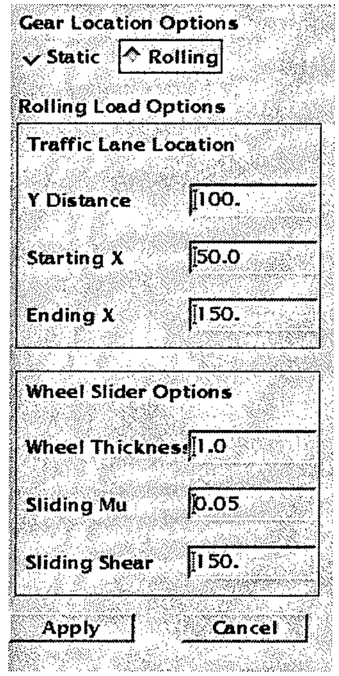


Figure 11. Gear location options form

“Sliding Shear” databox

Like the Sliding Mu, this is a field required by ABAQUS for contact surfaces. It is the shear stress limit for the contact surface. Its effect on the model is unknown.

“Apply” button

Apply will save the data entered on the form.

“Cancel” button

Cancel will close the form without updating the data that were changed.

Defining the Section of Pavement

A section of pavement is defined by its X and Y dimensions and by the number of layers and their thickness in the Z direction. The X dimension of the pavement is the dimension in the direction the plane would travel. The Y dimension is perpendicular to that. The origin of the model is at the aft starboard corner of the pavement, at the top surface. All of the model will be located at positive horizontal values, but most of the model is located at a negative Z coordinate location. Only the rolling load slider will be slightly positive.

The layers are specified from the top down, with layer 1 being the surface, and layer 2 or layer 3 being the bottom layer as shown in Figure 12. PMGI is set up to handle as many as five distinct layers.

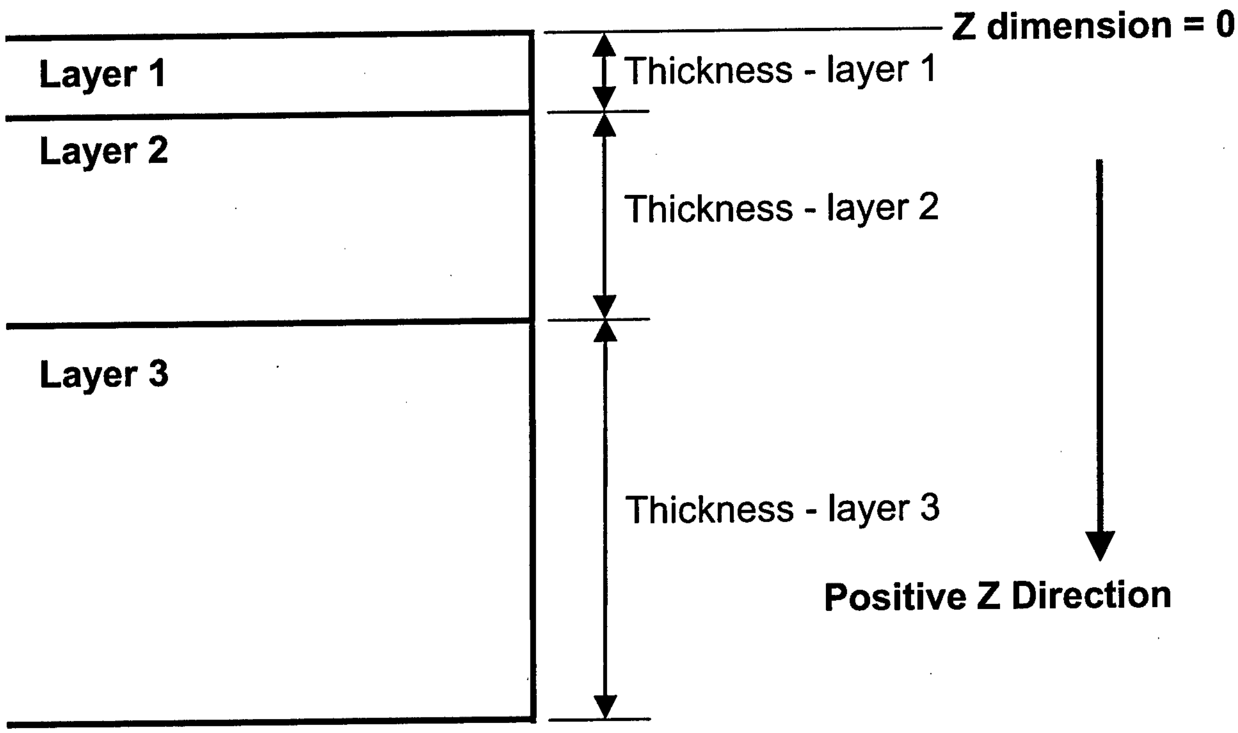


Figure 12. Typical layered system

The thickness of each layer can be specified independently. Additionally, the number of elements and transitions in the Z direction can be specified independently for each layer. When specifying a biased mesh, the positive direction for the elements will be downward. Thus when specifying L1 and L2, the L1 refers to the upper element size (nearest the top surface) and L2 to the lower (Figure 12).

When contact is specified between layers, the entire surface of each layer is made into a contact surface. The bottom of the upper layer is made the master surface; the top of the lower layer is the slave surface. In MSC/PATRAN, this would result in the master surface arrows pointing downward. This contact surface is specified to be small sliding, implying that any element is unlikely to slide, more than a distance of its size away from its initial location. The ABAQUS User's Manual presents more technical details of this interaction.

"Pavement Dimensions" Form

Layer options

The "Pavement Dimensions" form is shown in Figure 13.

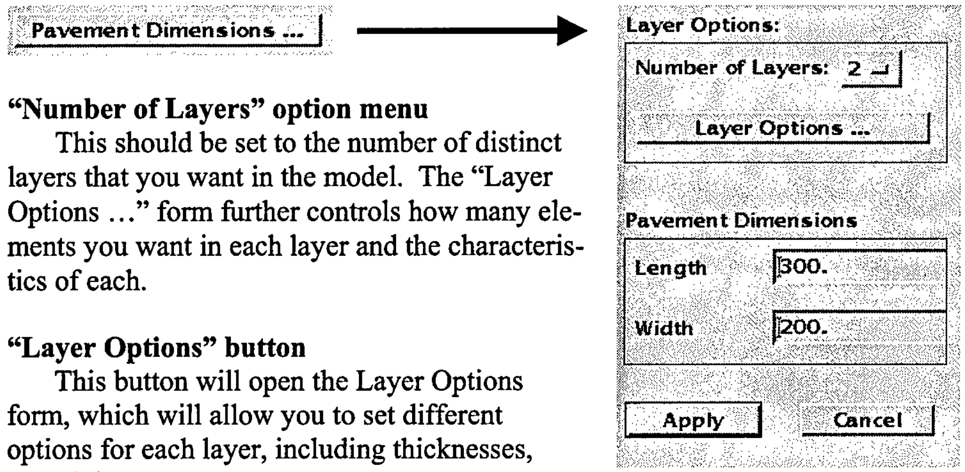


Figure 13. Pavement dimensions form

"Number of Layers" option menu

This should be set to the number of distinct layers that you want in the model. The "Layer Options ..." form further controls how many elements you want in each layer and the characteristics of each.

"Layer Options" button

This button will open the Layer Options form, which will allow you to set different options for each layer, including thicknesses, materials, contact, etc.

Pavement dimensions

"Length" databox

This is the length of the pavement in the X direction, the direction of the aircraft's travel. Remember to use units consistent with the wheel patterns and materials.

"Width" databox

This is the size of the pavement section perpendicular to the plane's direction. Remember to use consistent units.

"Apply" button

This will save the values entered on this form.

"Cancel" button

This will close the form without saving anything that was entered or changed.

"Layer Options" Subordinate Form

This form (Figure 14) will be updated to match the number of layers selected on the "Pavement Dimension" form. If a layer is added, it will be necessary to close this form and reopen it to force the update. A new layer will be added to the bottom of the list.

Layer options

There will be one row in this box for each layer in the mode as specified on the "Pavement Dimensions" form. Each layer has three main fields displayed at the right and three other fields that will be visible if the horizontal scroll bar is used.

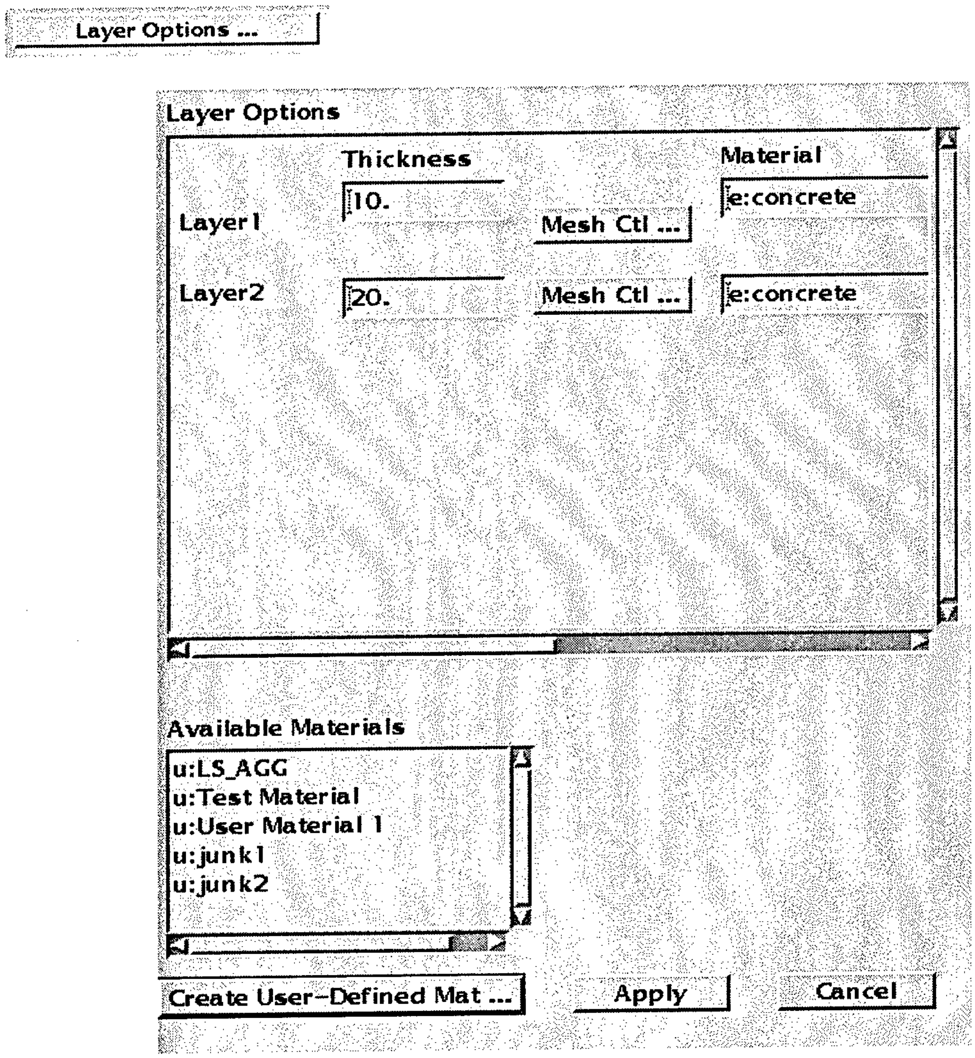


Figure 14. Layer options form

Main input fields

"Thickness" databox

This number is the thickness of the layer in question. Remember to use consistent units. This is a real number field.

"Mesh Ctl..." button

This button brings up the mesh-control form that allows you to specify the number of elements to use through the thickness of the specific layer, the length of the elements, if the mesh is to be transitioned to thicker elements, etc. Later in this chapter, a discussion concerning controlling the mesh in the vertical direction follows.

“Material” databox

This is the name of a material to use for the selected layer. Selecting a material from the “Available Materials” listbox will load that value into this box. If the material you want has not been defined, you can type the name into the databox manually. Material names in the databox will be prefixed with “p:”, “u:”, or “e:” to inform PMGI (and the user) where the model is defined:

a. “p:” designates a material defined in MSC/PATRAN prior to opening this form.

b. “u:” designates a material defined in a “.umat” file.

c. “e:” is a material that does not exist in either of these places. If PMGI is run using one of these materials, MSC/PATRAN will create a material with this name, but no properties will be assigned. If you type a material name into the databox that does not exist, PMGI will automatically append the “e:” to the name.

If the horizontal scroll bar is used, three more fields will appear for each layer (Figure 15). These fields control contact between the layers. By default, no contact is chosen. This means that layers share common nodes and are essentially rigidly connected together. If contact is desired, duplicate nodes will be created at the interface, and the frictional coefficient and shear stress limit for each contact surface needs to be specified.

“Cont Below” toggle

This toggle activates the contact options for the interface on the bottom side of the selected layer. If this is the bottom layer, the settings are ignored. Layers enabled with this toggle will not share nodes with the layer below, and a contact surface will be defined for both sides of the interface.

“Friction Coef” databox

The value input here corresponds to the value of the frictional coefficient, mu, defined on the ABAQUS *FRICTION card.

“Shear Limit” databox

This value corresponds to the shear stress limit specified as the TAUMAX parameter on the ABAQUS *FRICTION card.

The remaining widgets on the form are reasonably self-explanatory.

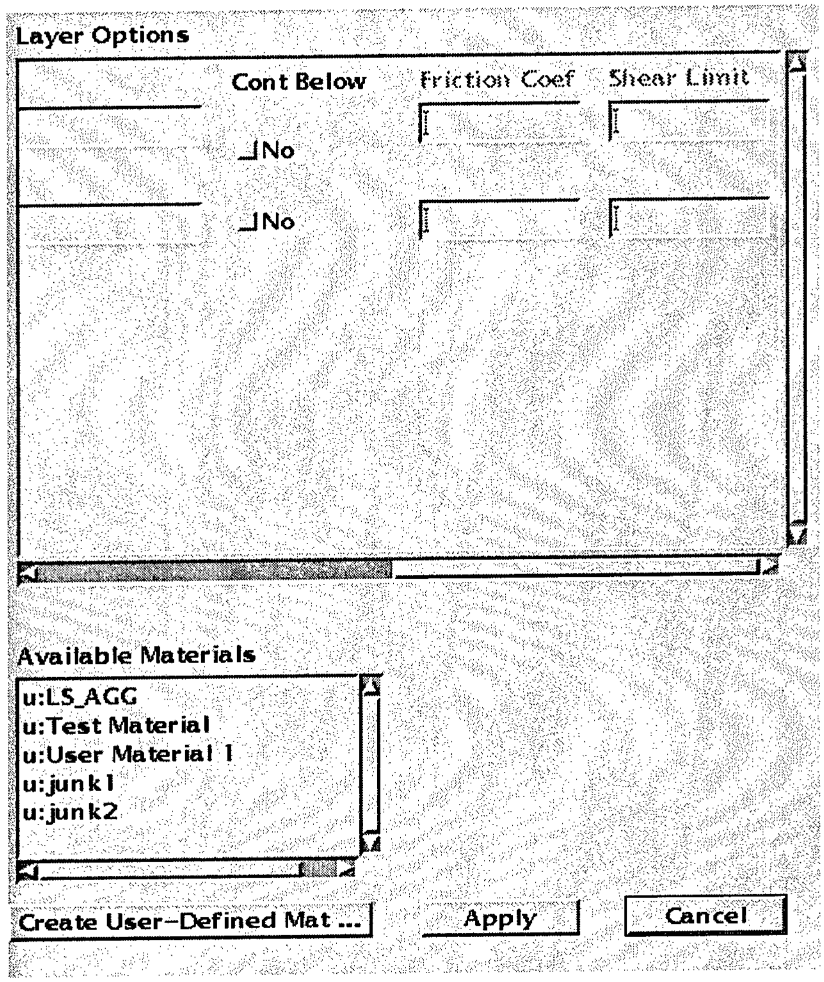


Figure 15. Material selection form

"Available Materials" listbox

This is a list of materials that have been defined in the MSC/PATRAN database, as well as a list of user-created materials defined in ".umat" files in the current directory. Picking a material in this list will load the name into the currently selected "Material" databox. If focus is not in a "Material" databox, selecting items in the box will have no effect.

"Create User-Defined Mat ..." button

This button will open the "New Material" form, allowing the user to define coefficients for a new material. After creation, the "Available Materials" listbox will be automatically updated to reflect the new addition.

"Apply" button

This will save the current data entered into this form.

"Cancel" button

This will close this form without saving any of the data entered or changed.

Meshing the Model

The basic meshing strategy used in PMGI is to create a two-dimensional (2-D) mesh at the top surface of the pavement section, then extrude the 2-D mesh downward into three-dimensional (3-D) elements. The user has a choice of 8-node linear solid elements, or 20-node quadratic elements.

For a static analysis, the model is meshed in three distinct sections:

a. The wheels.

b. The local area around the wheels.

c. The rest of the model.

For a rolling load the pavement is meshed in 10 distinct areas:

a. The wheel sliders.

b. The area where the wheels will move across.

c. Eight sections surrounding this area.

Options to control the mesh are located on the meshing options form (for controlling the 2-D mesh that gets extruded) or the layer options form (to control the mesh in the vertical dimension).

The "Meshing Options" Form

This form controls the mesh sizes and style on the model. It also lets you choose element types (Figure 16).

Solid element type

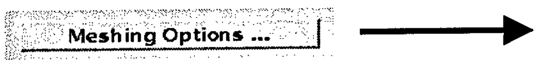
"Number of Nodes" switch

This switch lets you choose linear (8-node), quadratic (20-node), or full quadratic (27-node) elements. The 27-node option has not yet been implemented. Only one of these may be chosen, and it will be applied for the entire model, excluding the slider mesh for rolling loads.

Tire mesh

"Number of Elem in X" databox

This is the number of elements to use for the tire footprint in the X direction, in the direction of the aircraft's travel. This applies to the slider meshes for rolling loads as well.



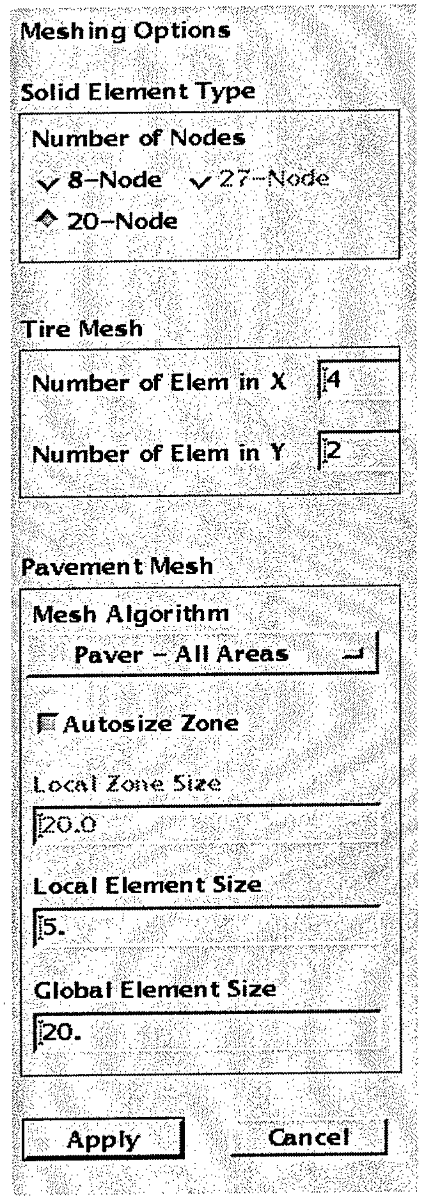


Figure 16. Mesh options form

"Number of Elem in Y" databox

This is the number of elements to use for the tire footprint in the Y direction, perpendicular to the aircraft's direction of travel.

Pavement mesh

"Mesh Algorithm" option menu

This menu lets the user choose the meshing algorithm for two of the mesh regions in the model. The model has three meshing regions:

a. The tire region – controlled completely by the "Tire Mesh" parameters.

b. The local region – a box enclosing the tire footprints but not necessarily extending to the edge of the pavement.

c. The global region – all of the model outside of the local region.

The mesh size for each of these regions can be controlled independently by the parameters in this box. Currently, the mapped mesher (IsoMesher) is used only for the tire footprints, and the free mesher (Paver) is used for the rest. As a result, this menu has only one selection for now.

When the "Rolling Load" option is used, the local region and sliders are meshed using the IsoMesher. The global region is meshed using Paver. This is because of the regular mesh desired for application of rolling loads.

"Autosize Zone" toggle

The local region encloses the area around the tire footprints. The local zone size determines how far the local zone extends beyond the maximum extent of the wheels. Enabling this toggle will set the local zone size to be 20 percent of the largest model dimension. If it is not toggled, the next box is activated.

"Local Zone Size" databox

If "Autosize Zone" is not enabled, the user must input a zone size into this databox. Note that the zone must be larger than zero, as the meshing algorithm is not set up to handle any other case. Note also, if the local zone is smaller than the local element size, a distorted mesh may result.

"Local Element Size" databox

This is the "Global Edge Length" parameter to be used when meshing local zones in the model. As mentioned before, this value should be smaller than the local zone size for a good mesh.

"Global Element Size" databox

This is the "Global Edge Length" parameter for the rest of the model. If this value is significantly larger (an order of magnitude or more) than the local element size, a messy transition may occur. Also, if the local zone approaches the edge of the model closer than the global element length, a distorted mesh may result in that area.

For a better explanation and some pictures of how these meshing parameters affect the model, refer to Chapter 5 for a description of the program mechanics.

"Apply" button

This saves the current form settings.

"Cancel" button

This closes the form without saving any of the changes or data entered.

Meshing the Wheels

The wheels (and wheel sliders for rolling loads) are meshed with a uniform mesh. The mesh is specified as "Number of Elements in X direction," and "Number of elements in Y direction." The MSC/PATRAN IsoMesher is used to generate the wheel mesh, resulting in a uniform mesh of rectangular elements. This same procedure is used for generating the slider modes for rolling load cases. In the static case, these 2-D elements are extruded downward to create 3-D pavement elements. In the rolling case, the 2-D elements are kept, and slid across the pavement model to mimic a rolling load. A mesh designated to have four elements in the X direction and two in the Y direction would appear as in Figure 17.

Meshing the rest of the model (static analysis)

The local area around the wheels is meshed using the MSC/PATRAN Paver with an element length determined by (a) the wheel meshes and (b) the local element length parameter. In other words, the mesh is matched along the edges of the wheels, and then transitioned elements are sized according to the local area length parameter. The size of the local area is defined by a "Boxsize" parameter. This parameter represents the distance from the outermost point on a wheel to the outer edge of the local zone (Figure 18).

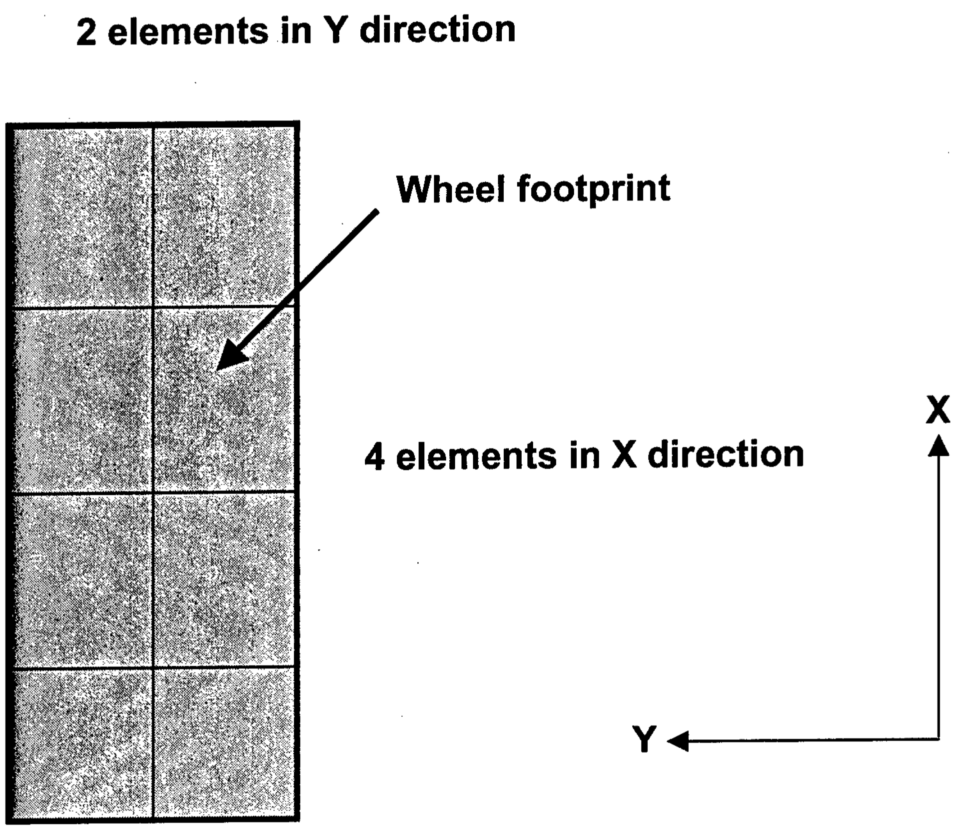


Figure 17. Wheel load diagram

The remainder of the model is meshed using the PAVER as well. The mesh size is determined first by the element size along the boundary of the local box(es) and then by a global edge-length parameter. This means that the mesh size is matched along the outer boundary of the local area box and then transitioned to the specified global area length.

This allows a regular mesh under the wheels, a fine mesh near the wheels, and a coarser mesh farther away from the wheels. By specifying the wheel mesh, the local element length, and the box size, the extent and fineness of the mesh in the area of interest can be arbitrarily controlled.

As each section is meshed, it is extruded downward. At the conclusion of the meshing, abutting areas (wheels/local area and local area/global area) within each layer are equivalenced, and layers that are bound together are equivalenced. Layers that are to be separated by contact surfaces are not equivalenced.

Meshing the rest of the model (rolling analysis)

The wheel sliders are independent of the rolling load model, so the local area is defined slightly differently. Like the static case, a local box is drawn around

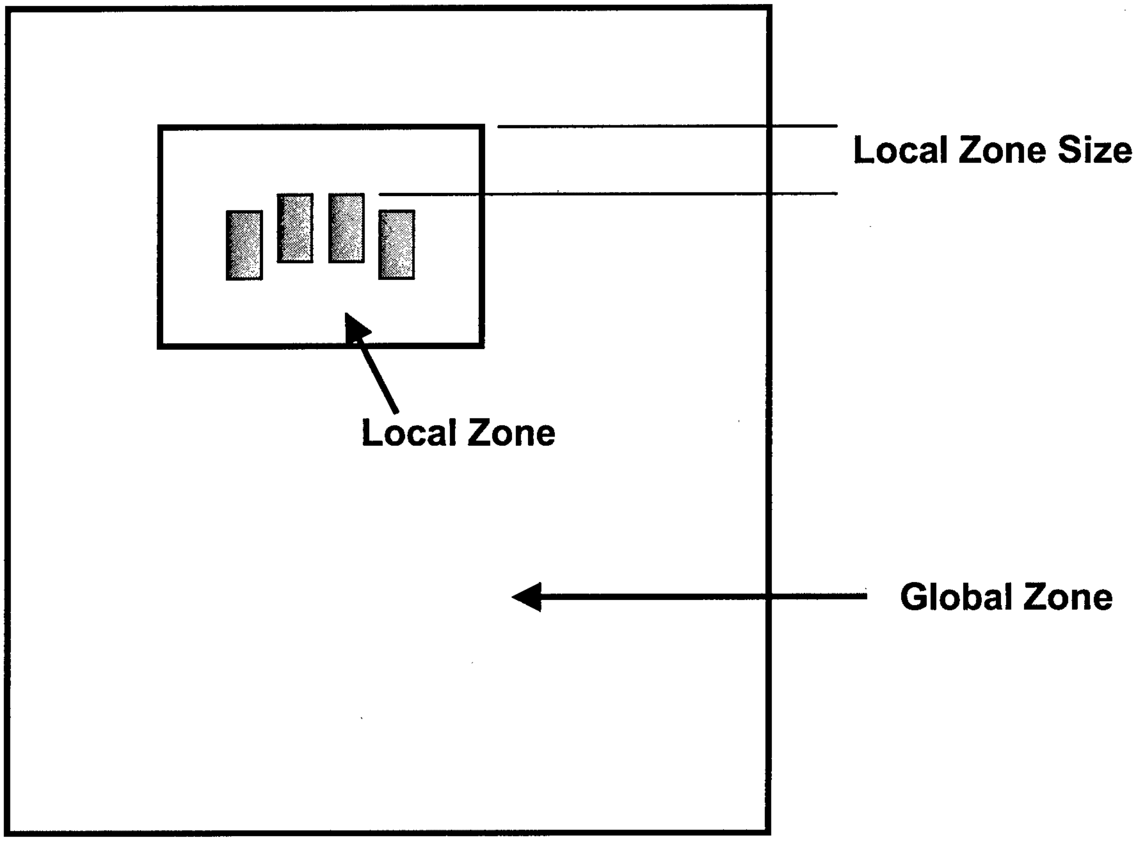


Figure 18. Surface meshing zones

the wheel sliders. For the rolling case, however, the box is extended to include the final position of the wheel pattern as well as the initial position. This extended box is the new "local area." Because the wheel sliders are independent of the pavement, it is not necessary for the pavement mesh to match the wheel footprints in any way. Therefore, the pavement is meshed with a uniform N X M mesh using the IsoMesher in the local areas, and the remainder of the model is meshed using the Paver. The remainder of the model is divided into eight sections that surround the local box. Each section is meshed independently, although using the same parameters. In some cases, where the load comes closer to the edge of the model than the global area length, the local section is meshed using the local length to force a better mesh. The model is divided as shown in Figure 19. Like the static loading, the surface mesh (with the exception of the wheel sliders) is extruded downward into 3-D elements. Equivalencing of coincident nodes takes place exactly like the static case.

Meshing in the vertical direction

The vertical mesh is controlled via the "Mesh Ctl..." buttons on the layer option form, since the vertical mesh is layer-specific. When the button is pushed for a layer, the standard MSC/PATRAN meshing control form appears. There are really several different versions of this form, depending on what type of mesh

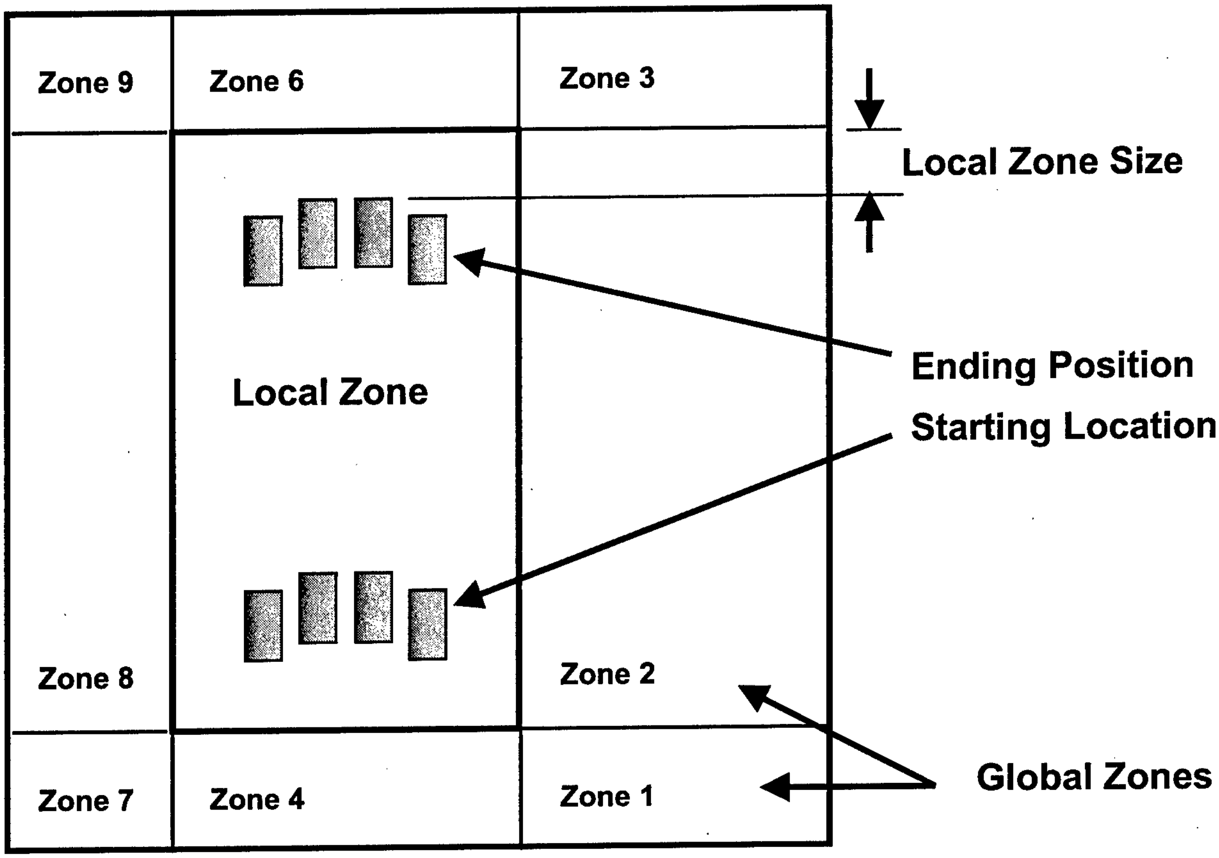


Figure 19. Global surface zones

transition is being considered. There are three basic forms depending on whether one wants:

a. A uniform mesh.

b. A one-way bias.

c. A two-way bias.

Note that the form settings only affect the mesh in the vertical direction. The mesh is extruded from the surface mesh all the way to the bottom of the model. It is only the size of the elements in the vertical direction that this form controls.

Uniform mesh

The two choices on this form (Figure 20) allow you to choose the number of elements through the thickness of the layer, or to choose the approximate length of the elements through the thickness.

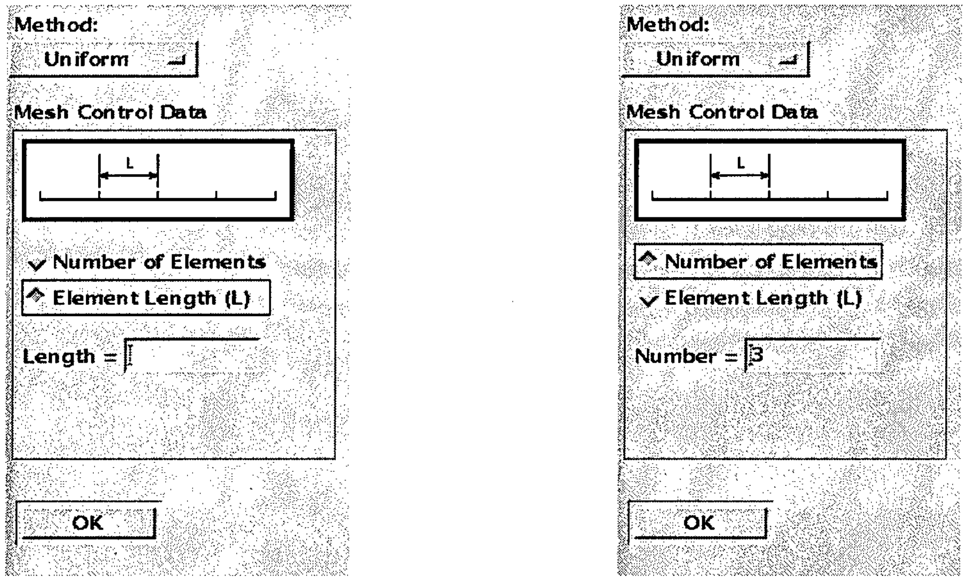


Figure 20. Uniform mesh

One-way bias

The two choices here are explained on the form by the two icons (Figure 21). Basically, you can choose the number of elements and a size ratio (letting MSC/ PATRAN choose the exact element sizes), or you can specify the exact sizes and let MSC/ PATRAN choose the number of elements.

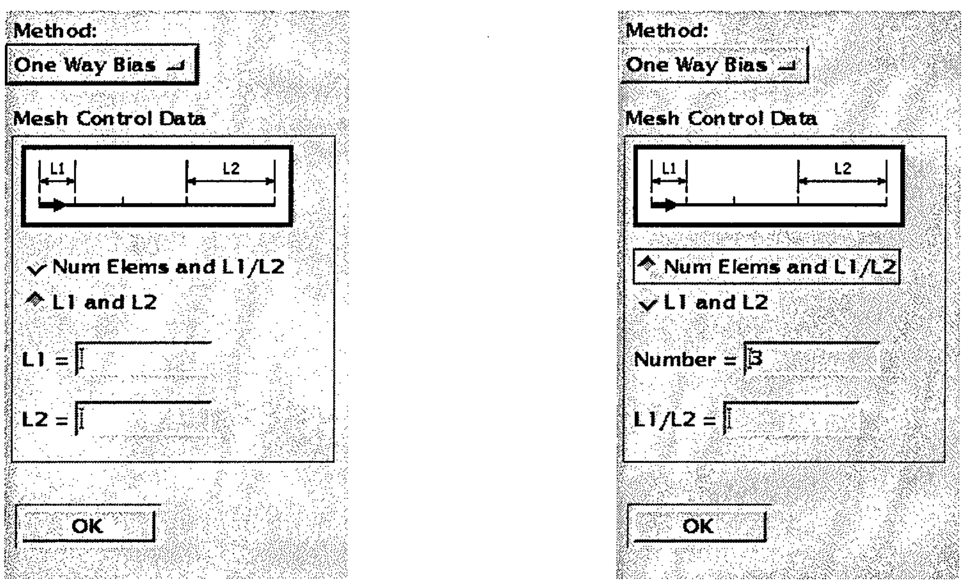


Figure 21. One-way bias mesh control form

Two-way bias

The choices here are similar to those for the one-way bias and explained by the icons (Figure 22).

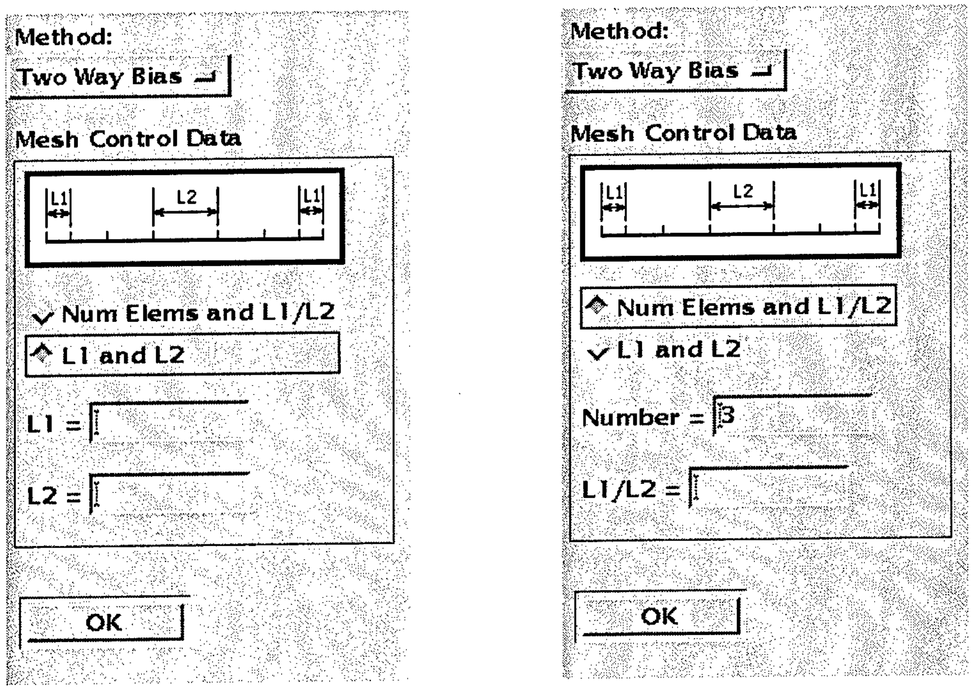


Figure 22. Two-way bias mesh control form

Note that the "OK" button on all of these mesh control forms hides the form. It functions as an apply button and does not provide for a cancel button. It is also identical to the same mesh control form in the MSC/PATRAN FEM application section.

Loading the Model

The loads on the model are indirectly determined by the specification of the wheel configurations and the type of loading. In the static case, the PRESS parameter in the ".gear" file determines what pressure load will be applied to each tire.

In the static case, this pressure is applied uniformly to the elements where the tires sit. This is the only loading condition applied to the model.

In the rolling case, the same is true, but the wheels are now 2-D shell elements sitting above the surface of the pavement. A preliminary ABAQUS step loads these sliders and closes the contact surface underneath them, putting the load onto the model. Then, an additional lateral displacement loading is applied to the sliders. By breaking the displacement loading into a series of small steps, the wheels effectively move across the surface, loaded with the pressure from the first load step.

There is no specific form to control the loading, as the pressure is determined by the choice of gear, and the displacement is determined by the starting and ending distances on the gear location form.

Boundary Conditions

PMGI currently supports two different types of boundary conditions for the mode:

a. Fixed

b. Springs

The fixed condition constrains all of the boundary nodes in the three translational directions. These displacement boundary conditions are divided into five separate MSC/PATRAN LBC sets to facilitate model verification and viewing. There is one set for each on the five faces of the pavement model: the front, back, two sides, and the bottom.

The sprung condition makes use of the ABAQUS *FOUNDATION capability. This capability allows specification of a stiffness/unit area across an element face. Unfortunately, this capability is not supported by MSC/PATRAN. Therefore, PMGI generates the *FOUNDATION cards directly and writes them to an external file that can be included into the ABAQUS deck manually. Alternatively, it can be included automatically by reading it into the Direct text box on the MSC/PATRAN Analysis form. In future versions of PATRAN, the *FOUNDATION data will be able to be directly loaded into this box.

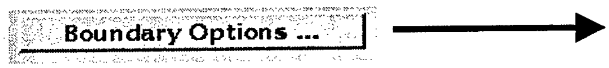
The "Boundary Options" Form

This form (Figure 23) has three sections that can be active depending on the settings of the three toggles. Three boundary conditions are available, and each section of the form has options that pertain to that boundary condition. Conditions will be applied uniformly to all of the nodes on the boundary of the model.

These three toggles, Fixed, Springs, and Infinite Elements, determine which boundary condition will be used on the model. Only one is allowed to be active at any time. When one is chosen, the options associated with that condition become active, and all of the others will be deactivated. The "Infinite Elements" option is not yet activated.

Fixed

There are no options for this condition. All nodes on the model boundary will be rigidly constrained in the three translational degrees of freedom.



Springs

"X Spring Stiffness" databox
"Y Spring Stiffness" databox
"Z Spring Stiffness" databox

The value in this box is sort of a stiffness per unit area. It is the value specified on the *FOUNDATION card in ABAQUS. The X springs are those on the faces of the model at constant X values; i.e., perpendicular to the aircraft's direction of travel. The Y springs are on the faces parallel to the aircraft's travel, at constant Y values. Z springs are on the bottom horizontal surface of the model.

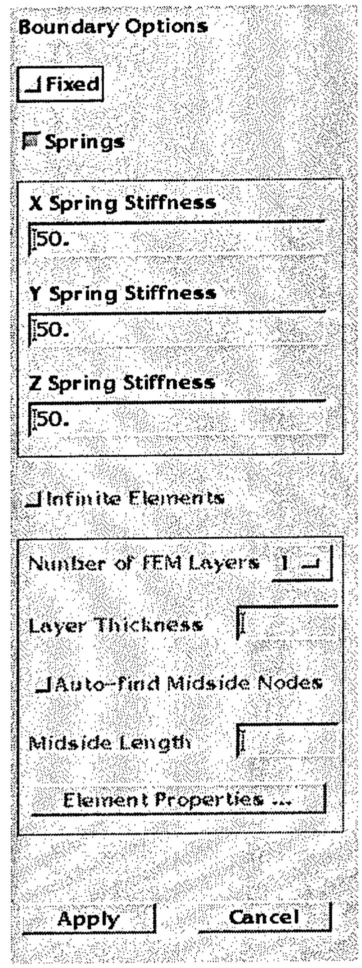


Figure 23. Boundary options form

Infinite elements

"Number of FEM Layers" options menu

This specifies the number of ordinary FEM elements that will be included between the model boundaries and the start of the infinite elements. It can be set from 0 to 4.

"Layer Thickness" databox

This is the total thickness of the above buffer of elements. Remember to use consistent units.

"Auto-find Midside Nodes" toggle

The midside node of an infinite element is supposed to be located such that the distance from the load to the face of the element is equal to the distance from the face of the element to the midside node. Activating this toggle will automatically calculate this distance for the five faces of the model.

"Midside Length" databox

If the previous toggle is not active, specify the distance from the element face to the midside node of the infinite elements. This option needs some work.

"Element Properties" button

This will bring up a form to input the properties of the infinite elements. This form has not been written yet.

"Apply" button

This simply saves the current form settings.

"Cancel" button

This closes the form without saving any changes or entries on the form.

5 The Internal Operation of the PMGI

The next two sections describe the steps that PMGI goes through to build a model. It is intended mainly as a reference so that the relevant MSC/ PATRAN function calls and procedures can be traced at a later date.

The Mechanics of Generating a Static Model

The mechanics of generating a static model are as follows:

a. Set the analysis preference to ABAQUS. Some of the material calls are ABAQUS specific and will generate errors if the analysis preference is set to anything else.

b. First, a few checks are performed on the data to eliminate some obvious problems. Some of these checks:

(1) The specified aircraft configuration and location does not place the aircraft off the pavement.

(2) The local region specified does not run over the edges of the model.

(3) Different gear configurations do not overlap each other.

(4) Different gear configurations do not approach each other too closely.

(5) Values have been specified for all required data.

(6) The material and gear files are complete and don't have internal conflicts.

c. Create curves defining the outer boundary of the pavement, controlled by the LENGTH and WIDTH values. The edges are created using the asm_create_curve_xyz function. These are then chained together into a loop using sgm_create_curve_chain_v1.

d. Compute the local box size if not specified by the BOXSIZE parameter. The default size is set in the create_model subroutine if the value passed to it is <0. Otherwise, the passed value is used. The box dimensions and location are defined by adding the value of box size to the maximum extent of the wheel pattern and subtracting it from the minimum extent.

e. Draw curves defining the local region boxes. These are processed similarly to the outer pavement boundary in Step c.

f. Draw curves defining each wheel footprint. Again, this is done just like the outer boundary in Step c.

g. Create a surface from each loop defining the wheels using sgm_create_surface_trimmed. This is created using the command to create a trimmed surface; however, it becomes a simple green surface because it is created from four bounding curves.

h. Seed the four edges of each curve based on the TIREX and TIREY parameters. This is done with the mesh_seed_create command.

i. Mesh the wheel footprints using the IsoMesh mapped mesher to achieve a uniform 2-D element mesh on the surface. The PCL routine is fem_create_mesh_surf_2.

j. Extrude the 2-D elements downward into 3-D hex elements. The distance and number of elements are controlled by the NUMEL and THICK parameters for each layer.

k. Assign a property set to each group of elements based on the MAT parameter for each layer. The property set will be named "tire_<aircraft number>_<wheel number>_<layer number>" So, for instance, wheel 1 of aircraft 2 in layer 3 would be named "tire_2_1_3."

l. Load the top layer of elements with the pressure loading specified by the PRESS value for this aircraft.

m. Create a trimmed surface from the local box (outer loop) and the wheels footprints (inner loops).

n. Mesh this surface using the Paver, since it will be an irregular area. The element size will be determined by the wheel mesh at the wheel boundaries, and by the LOCAL parameter elsewhere.

o. Extrude this mesh downward just like the tire meshes.

p. Assign properties just like the wheels, but the property set will be named "innerbox_<aircraft number>_<layer number>". (e.g., "innerbox_2_3").

q. Create a trimmed surface from the pavement boundary (outer loop) and the local regions (inner loops).

r. Mesh this surface using the paver, as it will also be an irregular surface.

s. Extrude this downward just like the other meshes.

t. Assign a property set called "outerbox_<layer number>" (e.g., "outerbox_3").

u. Equivalence nodes as appropriate:

(1) If there is no contact, equivalence everything.

(2) If there is contact, equivalence only within layers using the "list" option, creating groups for each layer.

(3) Equivalence between layers that do not have contact, using the "group" option.

v. Create a list of all nodes in the model that fall on the boundaries (sides and bottoms).

(1) For fixed boundary conditions, create a boundary displacement condition restraining all of these nodes in directions 1, 2, and 3.

(2) For spring boundary conditions, pass this list to a routine that writes out a *FOUNDATION card to an external file.

w. First set the analysis preference to ABAQUS for this modeler.

Rolling Loads

a. First set the analysis preference to ABAQUS for this modeler.

b. Find the maximum extent of the loading as defined by the pattern geometry, the XSTART, XEND, and YLOAD parameters.

c. Determine which of four possible scenarios exist for meshing (Figure 24).

d. Divide the pavement into a number of sections to accommodate these four arrangements.

e. Create curves to describe the outer boundaries of each of the required sections.

f. Create surfaces on each of these sections using the same trimmed surface command as for static loads. This will again result in simple green surfaces.

g. Create a surface mesh of 2-D elements on the local loading area.

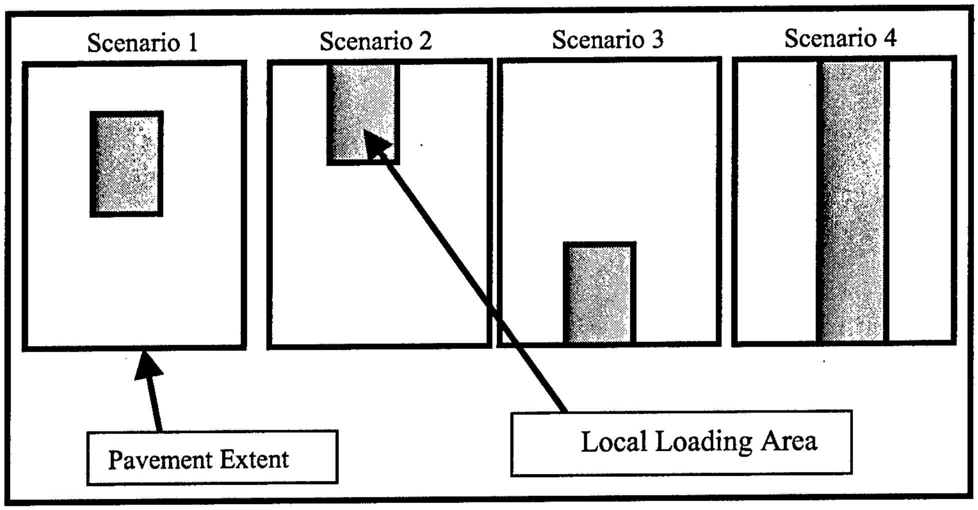


Figure 24. Meshing scenarios

h. Extrude the surface to form solid elements, following the layer information as before.

i. Assign a property set to this region, on a layer-by-layer basis, called "innerpart_<layer number>."

j. Create surface meshes on each of the other sections in turn. Use the GLOBAL length parameter for all of these, unless one is smaller than that, then use the LOCAL length parameter.

k. Extrude each of these surfaces down in turn to form solid elements.

l. Create and assign property sets to each of these solid regions, naming them "outerpart_<part number>_<layer number>" (e.g., "outerpart_6_2").

m. Equivalence all of the nodes within each layer, creating groups out of each layer at the same time.

n. Equivalence layers that have no contact between them.

o. Create curves that define the starting locations of the rolling tire prints.

p. Create surfaces from each of these loops.

q. Seed the curves to define a single element for each wheel print.

r. Mesh each surface with a single element.

Appendix A
Gear Description Files

Each landing gear configuration is described in a ".gear" file that describes the number of wheels, sizes, locations, etc. The filename must end with ".gear" for PMGI to recognize it. A typical ".gear" file might look like:

```
NAME     B-52 - Single Gear
WHEELS   4
WHEELXY  18.3    14.6
PRESS    234.0
COORD    1       1.00    -37.00
COORD    2       0.      0.00
COORD    3       0.      62.00
COORD    4       0.      99.00
```

There are five different "cards" in the file, each describing one aspect of the gear configuration. Each card is formatted in eight column fields, except for the NAME card. It is important that the keywords appear left justified on each card. Cards that do not have a recognized keyword are ignored. Order is not important. Each card and its format are described below.

NAME Card

Function:
Describes the gear configuration for PMGI. The description on this card is displayed in the different listboxes and spreadsheets on the different forms, not the file name of the ".gear" file.

Format:
A8, A40
NAME, Descriptive Title

Examples:

```
NAME    1234567890123456789012345678901234567890
NAME    C-130 Left Gear
NAME                   A LANDING GEAR
```

WHEELS Card

Function:
Tells how many wheels the configuration has.

Format:
A8,I8
WHEELS, Number of Wheels

Examples:

```
WHEELS  12345678
WHEELS  1
WHEELS         4
WHEELS           9
```

WHEELXY Card

Function:
Describes the dimensions of the tire footprint on the runway

Format:
A8, F8.x, F8.x
WHEELXY, x dimension, y dimension

Examples:
```
WHEELXY 1234567812345678
WHEELXY 10.34   15.56
WHEELXY 12      27
WHEELXY   11.7      24.6
```

PRESS Card

Function:
List the pressure load to be applied to each wheel footprint area.

Format:
A8, F8.x
PRESS, pressure loading

Examples:
```
PRESS   12345678
PRESS   10.5
PRESS     16
PRESS      136.7
```

COORD Card

Function:
To describe the locations of all of the wheels that comprise the configuration. There should be one COORD card to correspond to the number of wheels specified on the WHEELS card. The program does not check this thoroughly, so be careful specifying this.

Format:
A8, I8, F8.x, F8.x
COORD, wheel number, x location, y location

Examples:
```
COORD   123456781234567812345678
COORD   1       0.0     0.0
COORD      2    12        16
COORD          3    12.6    11.5
```

It may be desirable to model a full aircraft with multiple sets of landing gear on the field. This can be accomplished easily by duplicating the ".gear" file and giving each pattern a new name, such as:

```
NAME    C-130 - Port Gear
WHEELS  2
WHEELXY 22.4    17.84
PRESS   98.4
COORD   1       0.00    0.00
COORD   2       60.00   0.00
```

C-130P.gear

```
NAME    C-130 - Stbd Gear
WHEELS  2
WHEELXY 22.4    17.84
PRESS   98.4
COORD   1       0.00    0.00
COORD   2       60.00   0.00
```

C-130S.gear

Appendix B
Material Description Files

User-defined materials are described in ".umat" files. Like the gear description files, material files must end in ".umat" to be recognized by PMGI. Since the user-defined material interface has not been set up, current ".umat" files contain only one line and no material data:

NAME card

Function:
Provide a descriptive title to recognize the material by in the modeler.

Format:
A8, A40
NAME, descriptive title

Examples:

The "Create User-Defined Mat" Form

This lets you create ABAQUS *USER MATERIAL cards. It defines custom ABAQUS materials that are written to an external file, which can then be loaded into the direct text box on the MSC/PATRAN "Analysis" form. In this way, custom materials can be defined and included directly from the Pavement Modeler.

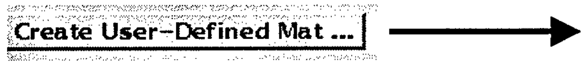
"Available Materials" listbox
This list is an echo of the listbox on the "Layer Options" form; however, it only contains user defined materials and has done away with the "u:" prefixes. It will be updated every time the form is opened, or any time the "Apply" button is pressed. It is presented here as a reference so that you will not create a duplicate material name. Selecting one of the materials on the list loads the material coefficients into the databoxes and allows editing of the user-defined materials.

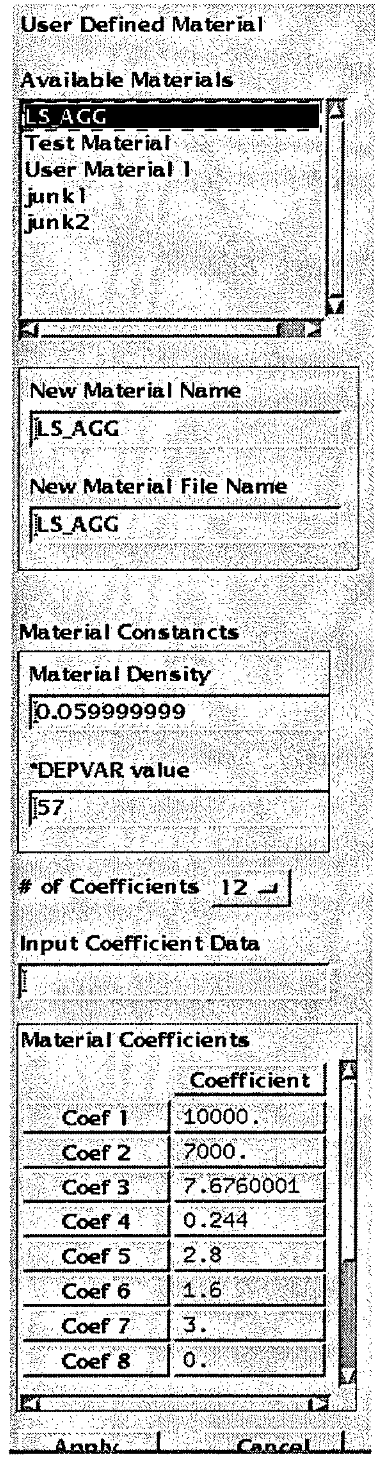
"New Material Name" databox
This is the material name that will be written to the NAME card in the ".umat" file. It is also the name that will appear in the "Available Materials" listboxes.

"New Material File" databox
This is the name of the ".umat" file that will be created when the "Apply" button is pressed. The suffix ".umat" will be automatically appended to the name specified in this box.

"Material Density" databox
This is the value that will appear on the *DENSITY card in the ABAQUS data deck.

"*DEPVAR value" databox
This is the value to appear on the *DEPVAR card in the ABAQUS deck.





"# of Coefficients" option menu

This tells PMGI how many boxes to present for input of the material coefficients and what number to put on the CONSTANTS parameter on the *USER MATERIAL card.

"Input Coefficient" databox

This is the spreadsheet data input box. When a spreadsheet cell is selected, the current value in the box is loaded into this box and highlighted. Pressing <CR> will load the value in this box into the spreadsheet of Material coefficients. It will also move the highlights box down one and load that value into the box.

"Material Coefficients" spreadsheet

This spreadsheet displays all of the currently defined material coefficients. Selecting a box of the spreadsheet loads the value into the Input Coefficients databox, as described above.

"Apply" button

Pressing this will write out a ".umat" file. It will check for the existence of the file first, allowing you to change the name if the file already exists. It will also create a ".dtf" file (of the same name) that will need to be read into the MSC/ PATRAN direct text input box on the Analysis setup form. Alternately, the ".dtf" file can be manually edited into the ABAQUS input deck.

"Cancel" button

This closes the form without making changes or writing files. Since PMGI does not read values directly off this form, there is nothing to save or restore.

Appendix C
Job Control Files

A job control file contains a description of all of the parameters that were used to generate a pavement model and its loading. Unlike the gear and material description files, the job control files (Figure C1) can be named anything the user desires. The extension ".jcf" files have been chosen for convenience, but it is not required to name them thusly.

```
NUMPL    1
PLANE    1        Sample Plane
WHEELS   1        2
WHEELXY  1        22.400   17.840
PRESS    1        98.400
COORD    1        1         0.000    0.000
COORD    1        2        60.000    0.000
LOADTYP  rolling
YLOAD    100.000
XSTART   50.000
XEND     150.000
WTHICK   1.
WFRICT   0.05
WSHEAR   150.
LOCATE   1        0.000    0.000
LAYERS   2
LENGTH   300.000
WIDTH    200.000
CONTACT  1        FALSE
THICK    1        10.000
FRICT    1        0.
STRESS   1        0.
MAT      1        e:concrete
TRANS    1        uniform
BIASTYP  1        number
NUMEL    1        3
L        1        0.5
L1       1        0.1
L2       1        0.2
L1L2     1        1.5
CONTACT  2        FALSE
THICK    2        20.000
FRICT    2        0.
STRESS   2        0.
MAT      2        e:concrete
TRANS    2        uniform
BIASTYP  2        number
NUMEL    2        3
L        2        0.5
L1       2        0.1
L2       2        0.2
L1L2     2        1.5
BOUND    springs
STIFF    50.      50.      50.
LAYERB   0
THICKB   0.000
MIDLEN   0.000
MESHTYP  20
TIREX    4
TIREY    2
MESHER   paver/all
LOCAL    5.000
GLOBAL   20.000
BOXSIZE  -1.000
```

Figure C1. Job form

Like the gear and material files, job control files are set up with keywords describing the parameter, and a number of eight-character fields that contain the values of the parameters. Note that all of the values contained in the gear files are stored in the job control file. This means that the gear file need not be present to run the job control file. It will, however, create the file if it does not exist. It will make up a name for the file, and load all of the appropriate information into it.

A typical job control is shown in Figure C1. Notice that the same keywords are used for the gear descriptions as are used in the gear description files.

The keywords not already described in the gear section (Appendix A) will be described on the following pages.

Like the gear and material files, order is not important. Any line that contains an unknown keyword will be ignored. The files may be edited to change a configuration or merely saved to provide a record of what was run.

Very little error checking is done when reading in these files. If a parameter is missing, misspelled, or wrong, it will probably be ignored.

NUMPL – number of aircraft in the model
Format:
A8, I8
NUMPL, number of aircraft

AIRCRAFT – NAME value of this aircraft
Format:
A8, I8, A40
AIRCRAFT, aircraft number, NAME of the aircraft

LOADTYP – loading type (static or rolling)
Format:
A8, A8
LOADTYP, loading type (choices are static or rolling – case is important)

YLOAD – Y location of gear pattern for rolling loads (ignored for static loads)
Format:
A8, F8.x
YLOAD, y distance to pattern

XSTART – Starting X location for rolling loads (ignored for static loads)
Format:
A8, F8.x
XSTART, starting x location

XEND – Ending X location for rolling loads (ignored for static loads)
Format:
A8, F8.x
XEND, ending x location

TIME – traverse time from x start to x end for rolling loads (ignored for static loads)

Format:
A8, F8.x
TIME, traverse time

LOCATE – X/Y location of gear pattern for static loading (ignored for rolling cases)

Format:
A8, I8, F8.x, F8.x
LOCATE, aircraft number, x location, y location

LAYERS – number of layers of pavement

Format:
A8, I8
LAYERS, number of layers

LENGTH – length of entire pavement section (X dimension)

Format:
A8, F8.x
LENGTH, x dimension

WIDTH – width of entire pavement section (Y dimension)

Format:
A8, F8.x
WIDTH, y dimension

NUMEL – number of elements through the thickness in this layer

Format:
A8, I8, I8
NUMEL, layer number, number of element through this layer

CONTACT – logical defining whether this layer has a contact surface on its bottom side

Format:
A8, I8, A8
CONTACT, layer number, contact flag (choices are TRUE or FALSE)

THICK – thickness of this layer

Format:
A8, I8, F8.x
THICK, layer number, thickness

REPORT DOCUMENTATION PAGE

Form Approved OMB No. 0704-0188

1. REPORT DATE *(DD-MM-YYYY)*	**2. REPORT TYPE**	**3. DATES COVERED** *(From - To)*
September 2003	Report 1 of a Series	

4. TITLE AND SUBTITLE

Pavement Model Generation Interface; Report 1, Formulation and Operation of the System

5a. CONTRACT NUMBER

5b. GRANT NUMBER

5c. PROGRAM ELEMENT NUMBER

6. AUTHOR(S)

Barton W. McPheeters, Donald M. Smith

5d. PROJECT NUMBER

5e. TASK NUMBER

5f. WORK UNIT NUMBER

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)

MSC-Software Corporation, 2 MacArthur Place, Santa Ana, CA 92707
U.S. Army Engineer Research and Development Center, Geotechnical and Structures Laboratory, 3909 Halls Ferry Road, Vicksburg, MS 39180-6199

8. PERFORMING ORGANIZATION REPORT NUMBER

ERDC/GSL TR-03-17

9. SPONSORING / MONITORING AGENCY NAME(S) AND ADDRESS(ES)

U.S. Army Corps of Engineers
Research, Development, Testing, and Evaluation Program
Washington, DC 20314-1000

10. SPONSOR/MONITOR'S ACRONYM(S)

11. SPONSOR/MONITOR'S REPORT NUMBER(S)

12. DISTRIBUTION / AVAILABILITY STATEMENT

Approved for public release; distribution is unlimited.

13. SUPPLEMENTARY NOTES

14. ABSTRACT

Today's pavement design calls for more emphasis on pavement performance prediction. This is a far more complex task than traditional designs that strive to provide safe thickness and specifications for the material. Modern computational mechanics provide application tools to deal with this new challenge, provided material models are created that predict cumulative deformations under traffic loading. The most theoretically rigorous analytical capabilities available today are found implemented within many of the commercial general-purpose finite element (FEM) computer programs. In order to correctly apply these analytical methods, it is necessary to produce FEM models of pavements systems that can be easily changed to allow for parameteric studies of the effects of load, environment, layer thickness, material properties, geometric properties and boundary conditions. The Pavement Model Generation Interface (PMGI) is a unique tool created expressly for that purpose. It allows pavement analysts to crate FEM grids and load steps for pavement systems with a minimal amount of effort when compared with the tedious procedures required to produce these massive 3-D grids by traditional preprocessing methods. This report documents the Formulation, Development, and Operation of the specific tools at hand, i.e., MASC/PATRAN and ABAQUS (A General Purpose Finite Element Code), in the development of an improved analytical model for pavements. This report will be the first of a series of reports documenting the Pavement Model Generation Interface (PMGI). The application and examples of using the PMGI will be detailed in subsequent reports.

15. SUBJECT TERMS

Finite element analysis	Pavements, flexible	Postprocessor
Moving wheel loads	Pavement layer interfaces	Preprocessor
	Pavements, rigid	

16. SECURITY CLASSIFICATION OF:			**17. LIMITATION OF ABSTRACT**	**18. NUMBER OF PAGES**	**19a. NAME OF RESPONSIBLE PERSON**
a. REPORT	**b. ABSTRACT**	**c. THIS PAGE**			**19b. TELEPHONE NUMBER** *(include area code)*
UNCLASSIFIED	UNCLASSIFIED	UNCLASSIFIED		57	