A COMPUTER-AIDED ANALYSIS SYSTEM WITH DBMS SUPPORT FOR FIBER-REINFORCED THICK COMPOSITE MATERIALS

Lisa K. Spainhour, Research Assistant
William J. Rasdorf, Associate Professor
Department of Civil Engineering
North Carolina State University
Raleigh, NC 27695

Edward M. Patton, Team Leader
Bruce P. Burns, Branch Chief
Mechanics and Structures Branch
Ballistic Research Laboratory
Aberdeen Proving Grounds, Aberdeen, MD 21005

Craig S. Collier, Engineer
Lockheed Engineering and Sciences Company
NASA Langley Program Office
Hampton, VA 23666

Abstract

The broad scope of the research described herein is the integration of several components of engineering software using a relational database. More specifically, a conceptual finite element material preprocessing system for fiber-reinforced composite materials was studied. In this computer-aided analysis (CAA) system, a materials database is integrated with several software components, including commercially available finite element analysis (FEA) programs and preprocessors, and tools for the design of laminated composite materials.

The focus of the system is on the integration of two- and three-dimensional composite materials data into several finite element analysis programs. Particular attention is given to analysis and design of components and structures using thick composite materials. Many engineering applications exist for thick composite structures; however, they have received less critical attention than the thin composite structures often used in aerospace applications. The primary objective of the composites analysis system is to enhance data transfer between and interaction among several engineering software programs with a minimum of user interaction.

This paper describes a specific implementation of a computer-aided analysis system that achieves this objective, detailing the need for the system and describing each of its components, including a composite materials database. The capabilities of the integrated system are discussed, including tasks such as composite laminate design, data entry, report generation, and interface file generation, performed in support of the finite element analysis capability. A major focus of the paper is on

the twofold role of the materials database in the analysis system, as both a passive data repository and as a dynamic data transfer mechanism. The use of interface programs and direct integration techniques are discussed in the context of passing materials data between the user and the database, and between the database and the various system components or application programs.

1 Introduction

Although a vast amount of software is available to solve many complex engineering problems, integration among the programs is often lacking. For example, the required file structure for batch input varies from program to program; therefore, the user must create multiple input files to manipulate the same data with different programs. In addition, there is no archival repository in which to store input information in a generic format. While there has been some focus on integration with respect to CAD/CAM software, relatively few standard programs exist to generate batch input files from a database. The problem is especially apparent in the realm of finite element analysis, where a large volume of input must be provided. In addition, the use of composite materials data for the analysis procedure further complicates the data entry task, because it magnifies the amount of data input required.

To address the problems of material data input and archival storage in the context of composite materials analysis, a prototype experimental, integrated finite element material property preprocessing system has been developed. The system focuses on the integration of two- and three-dimensional thick composite materials data with geometry and loading data to create an input file for finite element analysis programs, and is designed to provide simple data transfer among several programs with a minimum of user interaction. The computational system is composed of several stand alone, task-specific computer programs, some commercial and some written for this application. These programs perform tasks such as composite laminate design, data entry, interface file generation, and finite element analysis.

At the core of the system is a database, which provides an archive for fiber-reinforced composite material property data supporting finite element analysis. Components of the system request their data from the DBMS; where appropriate, output data is placed back into the database. Integration techniques aid in transferring data between the user and the database, and then between the database and the various application programs. The database, therefore, acts as a dynamic data transfer mechanism, as well as a passive data repository. This strategy enables the system to control the flow of information and allows the user to run the applications with a minimum of effort.

This paper provides some background information concerning the conceptual computer-aided analysis system, detailing the need for and benefits of such a system. The conceptual system is then illustrated through a detailed description of a personal computer implementation which focuses on the use of thick composites. The current system enables the user to generate materials data files for orthotropic materials, in either axisymmetric structures or two-dimensional structures subjected to membrane forces. Each of the required components is described, including the materials database, and a detailed example of system operations is provided. Finally, the work is summarized and conclusions are stated, including possible future directions of the work.

2 Composite Material Property Issues

It is readily apparent that, because the analysis system described herein addresses only the generation of material data for input to a FEA program, it cannot be considered to comprise a completely integrated finite element analysis system. It does, however, serve a purpose and fulfill a need in the engineering community. Because of analysis and design problems peculiar to the use of composite materials, generating material property values for program input is more difficult than with traditional engineering materials. Some of the very properties which make composite materials useful also make designing with them more difficult.

First, the complexity of the structure of the composite material itself must be considered. The stiffness or constitutive properties of fiber-matrix composites are usually orthotropic or even anisotropic. This factor is inherent in composite materials because of their design and manufacture. The fibers in a unidirectional ply of a fiber-reinforced composite are, by definition, aligned in a single direction, making the material orthotropic. The direction in which the fibers are aligned is known as the longitudinal direction, the axis across the fibers is known as transverse, and the third direction is through the thickness of the ply. To obtain the necessary three-dimensional composite material properties, the ply is often assumed to be transversely isotropic; that is, the properties in the two directions perpendicular to the fibers (transverse and thickness) are assumed to be identical.

Laminated composites, which are constructed by stacking several thin ply layers together, usually behave as either orthotropic or anisotropic materials. The material behaves orthotropically if the layers are balanced, with equal portions at positive and negative rotations, and/or symmetric through the thickness; otherwise, the behavior is anisotropic. If a complex laminate is designed, often no two coordinate axes share the same material properties. Furthermore, several different analysis techniques exist for calculating laminate properties, each giving slightly different results for the same problem. Many of these rely upon preliminary assumptions concerning ply properties, particularly transverse properties which are difficult to evaluate experimentally.

Because of the versatility that is inherent in manufacturing composites, a single composite structure may be designed and developed which has numerous ply combinations or stacking sequences. When this occurs, a greater amount of material property data has to be generated, because each laminate has a unique set of material properties. Two laminates with different stacking sequences that are designed from the same composite ply have different properties because the on-axis material properties of the laminate vary with each change in layup.

For these reasons and others which introduce even further uncertainty, the definition of material properties for a fiber-reinforced composite is very complex, and the process often has to be repeated numerous times for a single structure. In addition, design of structures using composite materials is a relatively new discipline for which few analytical tools exist. Strength properties of the material are often determined through experimentation rather than by calculation. For example, a particular layup may be chosen for a structure; a model is created and the analysis is performed; using the results of the analysis, a second layup is selected and the process is repeated. As a result, several different material configurations may actually be analyzed for a particular design scenario to determine the best design. We can conclude that designing a composite material structure is a repetitive process, well suited to automation.

It is evident, therefore, that there are numerous data collection and usage issues associated with composite materials, particularly in developing accurate property data for use in a FEA program.

We can make the following observations concerning composite materials.

- Composites have complex material properties—orthogonal or anisotropic.
- Each layup of a composite laminate has a different set of material properties.
- More than one layup may be used to design a single composite structure.
- Structural design with composite materials is generally by trial and error.

The key to designing with composite materials is obtaining complete and accurate data to predict the performance of the composite in a design scenario. An integrated computer-aided analysis system for fiber-reinforced composite materials can be designed to maintain material properties for both plies and laminates, and provide this data, either through direct integration or file transfers, to laminate design programs and finite element analysis programs. Data can be passed into and out of the integrated CAA system components as needed, and a top-level user interface allows many applications to be run with a minimum of user interaction.

3 A Computer-Aided Analysis System

The architecture of a prototype integrated computer-aided analysis system for fiber-reinforced composites, the Composites Database Interface (CDI) is described in this section. On a physical level, the primary purpose of the CDI analysis system is to automate the process of composite materials data generation and transfer to a finite element analysis system. A secondary purpose is to provide an archive for the storage of composite materials data. A third purpose of the research is to gain insight into the nature and organizational issues of the different types of data that are needed to define composite material properties. Existing technology in finite element analysis and database management is adopted to achieve the required functionality of the integrated system at the physical level.

Despite the fact that such integrated systems are not new in many domains, their introduction to the realm of composite materials analysis and design has not successfully occurred. This outcome is due in large part to the nature of the materials themselves and the overhead they bring to the development of a successful life-cycle model. The benefits of the system include the ability to generate a laminate in an ad hoc manner, having the material properties both immediately available for formatted FEA input and archived in a materials database.

To integrate the components of the Composites Database Interface, we use an incrementally expandable, or modular, architecture. The system components are embedded in an environment whose function is to facilitate data representation, data transfer, and data manipulation. The functional components of the CDI are described in Subsection 3.1 below, and the system integration issues are addressed in Subsection 3.2.

3.1 Functional Components

From a functional point of view, the CDI provides the following capabilities:

- Help,
- Data Manipulation,
- Laminate Design,
- Reports/Views, and
- FEA Interface File Generation.

These capabilities are provided by software components which are discussed in the following subsections and are illustrated in Figure 1. Certain components in Figure 1 are enclosed in dashed lines; these are the components illustrated in Section 4 below. The basic functionality and data requirements of each component are described. Taken as a whole, the functionality of the system components determine the features of the overall system, and the data requirements of each component specify the necessary schema and contents of the materials database. Note that the applications support only the data management aspects of the analysis system; the actual analysis capability is not provided in the CDI.

For each component, the necessary program code was written using the R:BASE For DOS application programming language; the CDI, therefore, can be executed only from within the R:BASE DBMS environment [1]. An external FORTRAN program, STIF3D, is imbedded within the laminate design component and invoked by R:BASE upon execution of the corresponding lower level system component. A top level menu-driven user interface allows the user to access the desired component to initiate an appropriate function.

In addition to the software components mentioned above, Figure 1 shows the following commercial stand alone applications: the R:BASE For DOS database management system, the ANSYS finite element analysis program, and the MAZE finite element analysis preprocessor. These components will not in themselves be discussed in detail in this paper. However, the composite materials database is discussed in Subsection 3.2, entitled *System Integration*.

3.1.1 Help Component

The purpose of the CDI Help component, as its name indicates, is to aid the user in executing the remaining CDI components. This functionality is consistent with the objectives of an integrated analysis system, as it reduces the user's dependence upon reference manuals and supplementary texts. The Help component provides the user with a collection of help screens, each describing a specific CDI function, providing general instructions about how it operates, supplying background information, and discussing possible options or command variations.

3.1.2 Data Manipulation Component

The function of the Data Manipulation component of the CDI is to provide the user with a simple means of entering, editing, and removing the data in the composite materials database, which acts as a data archive for the system. Two different methods for performing these actions are included in the CDI. The user can either manipulate the data by using on-screen data entry forms developed in

R:BASE, or enter the R:BASE Prompt by Example (PBE) mode directly to perform various tasks. In general, the forms and associated menu system guide the user, in a detailed step-by-step manner, through the data manipulation process; the PBE mode allows greater freedom but provides less guidance to the user.

Forms are data entry and modification tools that can be developed in R:BASE as part of a database definition. A form corresponds to a specific database table and contains numerous prompts. The data entered on each form corresponds to a single row in the table; each prompt, in turn, corresponds to a particular field of that row. When used for data entry, the fields of a form are initially empty. Data is entered into the form by typing a response to each prompt; a null value is entered by leaving the field empty. Once the data has been completely entered into a form and has been determined to be accurate, the user may enter it into the database as a new row in the corresponding table.

Forms can be used in a similar manner to modify data. The user first specifies which form and row (or rows) of data are to be modified. When the form appears on the screen, each field automatically contains the database values for the specified row. To change the values shown, the user simply types over the current data values, then selects the *Save* action from a menu. The user may also delete database rows by using similar menu options.

An alternate means of data entry and modification provided within the Data Manipulation component of the CDI is the Prompt By Example (PBE) mode of R:BASE. PBE allows the user to execute R:BASE commands directly, with the aid of screen prompts and context-sensitive informational messages. Possible PBE actions of interest are quite broad in their overall functionality and include modifying data, querying the database, using SQL commands, and accessing operating system utilities. Additionally, PBE commands may be used to read in data from flat ASCII files. This functionality is useful when a large volume of materials data is already on computer media; the DBMS has the capability to load the data directly into the appropriate database tables.

Most commonly needed operations can easily be performed using the appropriate PBE options. Certain actions, however, such as modifying the database structure, or haphazardly changing identification codes, should be avoided, as they will lead to inconsistent data within the database and among the CDI programs. Again, while the PBE mode allows greater freedom to the user, it also places more responsibility with the user for protecting the consistency and, therefore, the accuracy of the database data.

3.1.3 Laminate Design Component

The Laminate Design component of the CDI allows the user to design both quasi-isotropic and general laminates from the unidirectional ply data stored in the database. A laminate consists of layers of a single composite ply, oriented at appropriate angles to satisfy the expected loading configuration. A quasi-isotropic laminate, by definition, is one having equal ply portions at 0, 45, 90, and -45 degree angles; it is distinguished by equal elastic moduli in the major and minor directions. The general laminate may have plies oriented in any direction. The distinction between the two types of laminates is made because a simpler analysis technique is used in the CDI program for the design of quasi-isotropic laminates.

As input to the Laminate Design component, the user provides the name of the unidirectional

ply and the laminate code or stacking sequence for the layup to be designed. The laminate code indicates the orientation of each ply in the laminate. The material properties for the plies are obtained from the database. The laminate data, which includes stiffness properties and thermal expansion coefficients, is then generated and stored in the materials database. The process of designing laminates, therefore, can be separated conceptually into two phases. First the user designs the laminate by selecting the ply and stacking sequence, then the system analyzes the laminate and generates its material properties.

Two different models are available in the CDI for generating laminate stiffness data. Laminates may be analyzed using classical laminated plate theory, which is the current industry standard, as documented by Tsai in reference [2], by Halpin in [3], and by numerous other composite materials texts. Alternatively, a modified version of the STIF3D code, developed at the University of Delaware, may be used for laminate analysis. Thermal expansion coefficients can be generated only by using the classical laminated plate model. The remainder of this subsection focuses upon the classical plate model for laminate analysis.

Briefly, laminated plate theory, the most common analysis model, assumes a state of plane stress in the plane of the laminate. For each ply layer, a matrix of two-dimensional stiffness coefficients is computed from the elastic constants of the ply, its thickness, and its orientation with respect to an arbitrary in-plane axis. A matrix of normal stiffness properties, $[A]$, is generated for the laminate by taking the weighted averages of each set of ply properties.

Effective elastic constants can then be computed from the normal stiffness coefficients. This method, known as smearing, approximates the elastic properties as constants across the thickness of the laminate, giving effective rather than actual cross-sectional properties. Effective elastic constants are stored in the database to correspond to the input requirements of the integrated FEA codes. Had the codes required normal and bending stiffness coefficients, $[A]$ and $[D]$ matrices, these values would have been stored instead.

Because of the nonhomogeneous nature of laminated composites, coupling effects can normally be observed and calculated between the normal and bending stiffness coefficients of thin laminates which are not balanced and symmetric. As a result of these coupling terms, referred to as the $[B]$ matrix, the stretching or compressing of a laminate often results in bending and twisting deformation, and vice versa. However, because of the repetitive stacking sequences used for thick composite laminates, these secondary coupling effects are minimized, and the normal and bending stiffness may be considered separately [5] with a high degree of accuracy. This assumption would not be true, however, for thin laminates.

For laminate design using the classical laminate plate method, the required ply properties are the longitudinal and transverse elastic moduli, in-plane shear modulus, and major Poisson's ratio. The classical laminate model only generates two-dimensional stiffness data, however, and three dimensions of data are required for certain finite element analysis procedures. Properties in the third dimension must be approximated from the available ply and laminate data. A set of approximations that are based upon the assumptions of transversely isotropic plies and interpolation between known values are suggested to the user by the CDI program; the user may either accept these recommendations or modify the values as necessary. Note fields are provided in the database to document various assumptions made by the user.

To perform a thermal expansion analysis on a laminated composite, longitudinal and transverse

thermal expansion coefficients are required for the constitutive ply. Two-dimensional laminate thermal expansion coefficients are generated using a classical laminated plate model similar to that used for evaluating stiffness properties. Again, the assumption of transversely isotropic plies is used to obtain the thermal expansion coefficient in the third dimension. The user may elect to retain or modify this value.

3.1.4 Reports/Views Component

The Reports/Views component of the CDI provides the user with the capacity to observe the data stored in the database. Data may be viewed on the monitor or hard-copy reports may be generated. By means of menu selections, the user may specify the type and amount of data to be generated, thereby creating custom views and reports.

As with data entry and modification, the data represented by the collection of reports are the data contained in the material property database. Subsection 3.1.5 and the Appendix provide a description of those data items included in the database. In general, reports may be separated into two categories: identification reports and material property reports. Either category may be generated for both unidirectional plies and laminated composite materials.

An identification report provides general descriptive data about a composite. For a unidirectional ply, it includes the identification code used in the database and the composite, fiber, and matrix names, e.g., T300/5208, T300, and N5208, respectively. For a laminated composite, the same report lists the identification code, the composite name, and the laminate code, e.g., [0(2)/45/-45]4T. The user may elect to report on a single class of composites or to report on all available materials. For example, the user may report on all carbon fiber-reinforced composites (those with the class CFRP), or he may view all laminates defined in the database.

Material property reports are more detailed. In addition to the above information, they also include the values of various material properties for the selected plies or laminates, such as elastic and shear moduli, thermal expansion coefficients, Poisson's ratio(s), and density. Material property data may be either individually selected by composite name, or all available composites may be viewed.

3.1.5 Interface File Generation Component

The Interface File Generation component allows the user to create files of material data suitable for use with either the ANSYS finite element analysis program [6] or with the MAZE preprocessor for the NIKE2D and DYNA2D analysis codes [7]. Because of space considerations, this subsection focuses only on integration with ANSYS.

The ANSYS program encompasses three types of finite element analysis: two-dimensional, three-dimensional, and axisymmetric. The CDI system currently enables the user to generate materials data files for orthotropic materials, in either axisymmetric structures or two-dimensional structures subjected to plane stress or strain, that is, membrane forces. In ANSYS, the STIF42 element, a two-dimensional orthotropic isoparametric solid with two degrees of freedom per node, is used to model such structures. The two types of axisymmetric structures considered within the CDI are hoop-wound, in which the composite fibers are wrapped circumferentially around the

structure at some angle to the hoop axis, and radial composites, in which the fibers are aligned radially out from the center axis at some angle to the radial axis. The STIF42 element model requires elastic constants as input, e.g. E_x , ν_{xy} , etc., rather than stiffness coefficients ($[A]$, $[B]$, and $[D]$ coefficients).

The material properties required by ANSYS for analysis of two-dimensional plane structures subjected to plane stress in the X-Y plane are as follows: elastic moduli in the X and Y directions, minor Poisson's ratio in X-Y plane, shear modulus in X-Y plane, thermal expansion coefficients in the X and Y directions, and material density. For structures which are axisymmetric around the Y axis in the Z direction or for structures subjected to plane strain in the Z direction, three dimensions of material properties are required. In addition to the above properties, the elastic modulus and thermal expansion coefficient in the Z direction, and the minor Poisson's ratios in both the Y-Z and X-Z planes are required. Generic three-dimensional structures, which would require the same data set as axisymmetric structures, are not considered in the CDI.

As noted previously, the current scope of the materials database includes two-dimensional ply data (in the plane of the ply) and three-dimensional laminate properties. When three dimensions of ply properties are required by the FEA program, the plies are assumed to be transversely isotropic; that is, material properties in all planes perpendicular to the composite fiber are assumed equivalent. This data is not stored in the database to reduce redundancy. The challenge, therefore, is to correctly apply the two- or three-dimensional composite materials data to a two- or three-dimensional structural model. Conceptually, this procedure involves transforming the material properties from the axes of the composite (longitudinal and transverse for a ply, zero and ninety degrees for a laminate) to the global axes of the finite element model. Physically, this process corresponds to orienting the composite appropriately on the structure represented by the model.

Recall that, since the interface file generated by the CDI system only contains materials data, a separate file with the remaining FEA input data must also be developed using a text editor. This file, known as the batch input file, contains the finite element mesh, loading information, and the applicable constitutive relationships for the structure to be analyzed. These two files, the batch input file and the material data file, must be integrated together and used as batch input for the finite element program of choice. The integration of the material data file and the batch input file with the analysis code is handled differently for each FEA program integrated with the CDI.

Figure 2 shows a schematic representation of the data flow for integrating a set of data into the ANSYS FEA code. When the ANSYS Interface File Generation component of the CDI is executed, the materials data for one or more composites is placed into the material data file shown in Figure 2, in a format appropriate for ANSYS execution. Meanwhile, the batch file is created and, along with the mesh and loading data, several commands for directly accessing the materials data file are included. Therefore, the batch input file is processed in ANSYS until a command is executed transferring control to a specific line in the materials data file. The materials data file is then executed until a command returning control to the batch input file is encountered.

This method of integration is possible through a combination of several factors. First, the materials data file is formatted by the CDI program in such a way that it is equivalent to an ANSYS User file. A User file is a specially prepared file from which ANSYS can accept input and into which the batch file can pass arguments. The User file acts as a library file to the batch input file, which means that, once created, it can be used with multiple batch input files. Second, the batch input file must contain the required commands to properly reference the file and the

appropriate materials data within. In this way, the batch input file and the material data file are used independently to integrate the materials data with ANSYS.

To execute the ANSYS Interface File Generation component of the CDI, the user must provide the name of the materials data file, and for each material definition to be extracted from the database, an ANSYS data block name, the material name or identification code, and its orientation with respect to the ANSYS global coordinate axes. The material properties are then transformed from the orthotropic material axes to the ANSYS coordinate axes. Because the system is currently limited to two-dimensional coordinate transformations, the user will be prompted to approximate certain properties through the thickness of the laminate if he is performing an axisymmetric analysis. As with the Laminate Design component, the program offers suggestions to the user based upon the assumption of transversely isotropic plies and interpolation.

3.2 System Integration

The functional components of the CDI are integrated via the R:BASE relational database management system. A brief description of R:BASE is presented in Subsection 3.2.1 and the related integration issues are addressed in Subsection 3.2.2.

3.2.1 R:BASE For DOS

R:BASE For DOS, generally referred to as R:BASE, was developed by Microrim, Inc., and is a multi-user relational database management system that was designed for use on a microcomputer [1]. As R:BASE was created by one of the developers of the RIM DBMS, it retains much of RIM's functionality, supporting the integer, real and double precision data types, which are essential to support engineering applications [8]. In addition, R:BASE enables the user to interactively define database schemas, create custom data entry forms, and generate advanced output reports. The DBMS provides an advanced SQL-based command language and an application programming language. R:BASE is very sophisticated yet simple to use, primarily because of its data entry, manipulation, and report generation functions.

3.2.2 Component Integration

The role of R:BASE in the CDI is two-fold. First, R:BASE is used as a data transfer mechanism. This capability facilitates the transfer of data between the user and the system components, and it also allows data transfer among the set of heterogeneous application programs with a minimum of user interaction. R:BASE assumes a dynamic role in the evolving analysis process and provides the required data to each application program throughout the process. Second, R:BASE is used as a passive data repository. This capability provides the overall system with a centralized and manageable storage area for capturing the information obtained throughout the composite material life cycle. The data integrity of the process can thus more easily be monitored, because the information is maintained in one centralized location [9].

To transfer data between independent application programs, one needs to interface the corresponding programs. In an integrated system, two components are said to be interfaced if a data

path exists between the components. A data path can be created in one of two ways, either through direct integration or by means of data files. In direct integration, the data which is output as a result of executing one component is fed directly into a subsequent component as input. This situation occurs in executing most of the CDI application programs, because input data for the components is requested directly from the database and output data is placed back into the appropriate tables.

The second means of integrating two components is through the use of data files as interface files. With this method, an interface program maps the output data from one component to the input data format needed by another program. A data file is used in the CDI to integrate materials data into the finite element analysis codes. The Interface File Generation component of the CDI creates this data file in a format suitable for input into one of the FEA programs integrated with the system.

To create an integrated software system, such as the CDI, all components in the system must be interfaced with one another. We elected to interface each of the application programs with the materials database, effectively interfacing all of the components together. A benefit of this integration scheme is the ease with which interface programs can be developed for an R:BASE database. The integration procedure is simplified through use of the DBMS Application Programming Language, which provides commands allowing direct communication and interaction with the database.

Each interface program maps the data as needed by or received from its corresponding application program to a set of appropriate database relations in R:BASE. The collection of all the relations in R:BASE represents the data transmitted between the functional components of the CDI throughout the analysis process. Recall that Figure 1 depicts the CDI application programs, the interfaces, the centralized database management system, and the data paths between them.

To provide the necessary composite materials data in R:BASE to support the prototype, four initial relations have been developed. Sufficient relations have been defined to represent the data needed by the FEA programs for two-dimensional and axisymmetric analysis, as described in Subsection 3.1.5. Additionally, these relations are designed to support identification and material property data for unidirectional plies and laminated composites. A system of identification codes ties the data in the various relations together. The appendix includes a detailed description of the database schema, including table and column descriptions.

4 System Demonstration

This section includes a detailed transcript of a sample execution of the various components of the CDI system. For clarity, the following typographical conventions are employed in this section. Typewriter font (`typewriter`) is used to designate any text that is viewed on the monitor, including both prompts by the system and text entered by the user. In addition, an italics font (*italics*) is used to represent any CDI menu name or menu item. The example assumes that the user has initiated R:BASE and is currently in the R:BASE command mode, signified by an `R>` prompt on the screen.

4.1 Problem Statement

To illustrate the functionality of the CDI system, the design of the cylindrical sides of a large tank is considered. The tank, which is illustrated in Figure 3, is to be used as a pressure vessel. Accordingly, the loading on the tank consists of internal pressure, which induces only membrane forces into the cylindrical portion of the structure. Further, the load in the hoop direction is twice that in the axial direction. Because of the potential weight savings and a concern about a corrosive environment and thermally induced stresses, the tank is to be designed of thick graphite fiber-reinforced composite, which has a relatively small coefficient of thermal expansion. To best resist the pressure loading, the material will be hoop-wound about a form, creating a laminate which has twice the strength in the hoop direction as in the axial direction.

The tank designer wishes to perform finite element analysis, using ANSYS, on this fiber-reinforced tank. The first laminate configuration to be tested is a [30/-30/0/45/-45]5S layup. Figure 4 illustrates a simplistic finite element model of the tank, showing how the laminate axes correspond to the ANSYS coordinate axes. Because the tank only undergoes membrane forces, the ANSYS STIF42 element is an acceptable element with which to analyze the finite element model.

The composite material to be used in this design, identified as IM7/8551-7, is made from the 8551-7 epoxy matrix and the IM7 graphite fiber, and has the following unidirectional properties:

Elastic modulus in fiber direction (E_1): 23.0×10^6 psi
Elastic modulus in transverse direction (E_2): 1.21×10^6 psi
Shear modulus of elasticity (G_{12}): 0.83×10^6 psi
Major Poisson's ratio (ν_{12}): 0.34
Density (ρ): 148×10^{-6} lbf-s²/in
Major thermal expansion coef. (α_x): 0.24×10^{-6} in/in-°C
Minor thermal expansion coef. (α_y): 14.0×10^{-6} in/in-°C
Ply thickness: .0052 inches

These ply properties must be stored in the database using the Data Manipulation component of the CDI. The tank is composed of a laminate, however; as a result, the laminate property data must be generated and stored in the database using one of the models in the Laminate Design component. Finally, using the Interface File Generation component, the user must create material data files suitable for use with ANSYS. This entire procedure is illustrated in the subsections which follow.

The user may also edit or delete any database data with the Data Manipulation component, access Help, or print reports using the Report/Views component. Additional laminate design models exist, and interface files can also be generated for use with MAZE. Because of space considerations, these and other actions are not illustrated in this sample transcript.

4.2 Entry of Ply Data into Database

To initiate the Composites Database Interface, the user types the following command at the R> prompt: RUN COMP.CMD. An introductory screen first appears; the main menu, shown in Figure 5, appears after any key is pressed. From this menu, the user may access any available program action by selecting the appropriate option.

To enter the requisite ply data, the user selects the *Database* option from the main menu, then selects the *Enter Database Data* from the subsequent menu, *Database Actions*, illustrated in Figure 6. The last menu, *Enter Database Data*, then appears. From this menu the user selects the first option to enter data on unidirectional composites. This menu is illustrated in Figure 7. Subsequently, several prompts appear, which are shown below along with the appropriate responses.

```
ENTER THE COMPOSITE'S NAME: IM7/8551-7
ENTER AN IDENTIFICATION CODE IN THIS FORM: U1, U2.
(TYPE [ENTER] TO LIST CURRENTLY USED ID CODES): U5
```

Upon properly entering both the composite name and an arbitrary identification code, a data entry form appears on the screen. This form corresponds to the UNICOMP table, and contains ply identification data. The user then fills in the empty fields such that the form resembles the illustration shown in Figure 8.

After all data items have been entered, the form is saved, entering this row of data into the database table. A short pause will follow, then a second data entry form appears, corresponding to the UNIPROP table. The property data values given previously should be typed into the corresponding empty fields on this form. When complete, the form should resemble the illustration in Figure 9.

The ply identification and property data is now stored in the database. If necessary, the user may edit the data in these or any other database tables using the second option of the *Database Actions* menu. The data may also be viewed, printed, or used to create interface files for use with ANSYS or MAZE.

4.3 Design of Composite Laminate Using Ply Data

To design a laminate and generate its material properties from the corresponding unidirectional composite, the *Laminate* option is selected from the CDI main menu in Figure 5. The *Laminate Design* menu is then displayed, as in Figure 10. From this menu, the first option, *Classical laminated stiffness theory (Tsai)* is selected to develop a set of stiffness properties using classical laminate theory. Recall that we wish to design a laminate from the previously entered ply data with the stacking sequence [30/-30/0/45/-45]5S.

Following is a transcript depicting the required interaction in designing this laminate. Most queries are self-explanatory; however, comments have been interspersed where deemed necessary.

```
ENTER THE NAME OF THE UNIDIRECTIONAL COMPOSITE IN THIS LAMINATE.
(TYPE [ENTER] TO LIST ALL UNIDIRECTIONAL COMPOSITES): IM7/8551-7
```

```
DO YOU WISH TO DESIGN A QUASI-ISOTROPIC LAMINATE? (Y/N) N
```

```
ENTER THE LAMINATE CODE (STACKING SEQUENCE) IN THIS FORM:
```

```
[dir(layers)/dir(layers)...]XX,
```

```
WHERE (layers) IS OPTIONAL FOR ANY DIRECTION AND "XX" MAY BE
```

#T, FOR REPEAT SUBLAMINATE # TIMES,
 #S, FOR REPEAT # TIMES THEN MAKE SYMMETRIC,
 S#, FOR MAKE SYMMETRIC THEN REPEAT # TIMES.
 [30/-30/0/45/-45]5S

ENTER AN ID CODE FOR THE LAMINATE IN THIS FORM: L1, L2.
 (TYPE [ENTER] TO LIST ALL CURRENTLY USED ID CODES): L1

Recall that, for classical laminate method, certain three-dimensional laminate properties must be approximated. Suggestions are provided to the user, based upon the assumption of transversely isotropic plies and interpolation between known values. In this sample transcript, all default values are accepted by the user except those for the Poisson's ratios through the thickness.

APPROXIMATING REMAINING PROPERTIES...

TYPE [ENTER] TO ACCEPT OR TYPE A MODIFIED VALUE TO CHANGE.

Ez = E2 = 1210000. MODIFY? [N] [ENTER]
 ExBend = Ex = 9361206. MODIFY? [N] [ENTER]
 EyBend = Ey = 3059484. MODIFY? [N] [ENTER]

ENTER AN APPROXIMATION FOR NUyz AND NUxz BASED UPON NU12 = .34
 AND THE LAMINATE CODE = [30/-30/0/45/-45]5S

SHEAR MODULI WILL BE COMPUTED BASED UPON THESE APPROXIMATIONS.

NUyz? [.4] .45
 NUxz? [.4] .1
 Gyz = 417241. MODIFY? [N] [ENTER]
 Gxz = 550000. MODIFY? [N] [ENTER]

RECORD ANY NECESSARY DATA PERTAINING TO THE MODIFIED ASSUMPTIONS
 OR TYPE [ENTER] TO INDICATE THAT THE DEFAULT APPROXIMATIONS WERE USED.
 >NUyz = .45 and NUxz = .1, otherwise standard assumptions were used.

MATERIAL STIFFNESS PROPERTIES ARE AS FOLLOWS:

Ex = 9361206. Ey = 3059484. Ez = 1210000.
 ExBend = 9361206. EyBend = 3059484.
 Gxy = 4366850. Gyz = 417241. Gxz = 550000.
 NUxy = .928 NUyz = .45 NUxz = .1

WHERE "X" IS THE MAJOR AXIS (0 ROTATION) AND "Y" IS THE MINOR.

DATA FOR THIS LAMINATE IS NOW LOADED INTO THE DATABASE.
 REPEAT USING ANOTHER STACKING SEQUENCE WITH THIS PLY? (Y/N) N

REPEAT USING ANOTHER UNIDIRECTIONAL PLY? (Y/N) N

The material property data for this laminate is now stored in the database and may be edited, deleted, viewed, or used to create material data files.

4.4 Generating Material Property Files for ANSYS

To create a data file for transferring material property data to a FEA program, the user selects the *Files* option of the CDI main menu. The *Generate Interface Files* menu, illustrated in Figure 11, then appears, enumerating the types of data file which can be generated. Selecting the *Ingrid* option will initiate no action. This option indicates the future possibility of integrating additional FEA programs with the system, including the INGRID program. The ANSYS model will be demonstrated in this example. Similar steps are involved in creating files suitable for use with MAZE. A sample material data file in the format appropriate for ANSYS input is illustrated at the end of this subsection in Figure 12.

Following is a sample transcript of the user interaction involved in creating the material property data file. Recall that the structure we are modeling is an axisymmetric cylinder, that the composite is hoop-wound, and that the major laminate axis exactly corresponds to the ANSYS Z-axis (hoop). All of this information will be used in creating the material data file.

ENTER A NAME FOR THE ANSYS USER FILE: ANSYS.SPL

HOW MANY UNIDIRECTIONALS TO BE DEFINED? 0

HOW MANY LAMINATES TO BE DEFINED? 1

ENTER THE ID CODE OF LAMINATED COMPOSITE NUMBER 1 OF 1...

(TYPE [ENTER] TO LIST THOSE CURRENTLY DEFINED): L1

At this point, a list of all stiffness runs associated with this composite is printed. Recall that, there is more than one laminate design model and because each model may be run with different assumptions, more than one set of stiffness data may be defined for a single laminate. To complete the interface file generation, the user must select the appropriate data set; in this case, the user selects the first run to use the classical laminate data generated previously.

LAMINATED COMPOSITE MATERIALS

=====

LAM ID	LAMINATE CODE	UNI. COMPOSITE	METHOD	RUN ID
=====	=====	=====	=====	=====
				(STIFFNESS DATA ONLY)
L1	[30/-30/0/45/-45]5S	IM7/8551-7	CLASSIC	1
			STIF3D	2

ENTER THE DESIRED RUN ID NUMBER: 1

ENTER A DATA BLOCK NAME FOR THIS MATERIAL (UP TO EIGHT CHARACTERS):

TANK1

TYPE [P] IF DEFINING A 2-DIMENSIONAL PLANE SOLID IN THE
X-Y PLANE; TYPE [A] IF DEFINING AN AXISYMMETRIC CYLINDER WITH
SYMMETRY AROUND THE Y-AXIS IN THE Z-DIRECTION. A

TYPE [H] IF THE COMPOSITE IS HOOP-WOUND; TYPE [R] FOR RADIAL. H

ENTER ANGLE BETWEEN COMPOSITE'S MAJOR DIRECTION AND ANSYS Z-AXIS: 0

The reader should note that creating a material data block for a two-dimensional plane structure is very similar to the above. The user is prompted to enter the angle between the composite's major direction and the ANSYS X-axis, and the previous query, being irrelevant, is omitted.

CREATING ANSYS DATA BLOCK...

APPROXIMATING REMAINING PROPERTIES...

TYPE [ENTER] TO ACCEPT OR TYPE A MODIFIED VALUE TO CHANGE.

G hoop-axial = 4366850. MODIFY? [N] [ENTER]

NU axial-radial = .45 MODIFY? [N] [ENTER]

NU hoop-radial = .1 MODIFY? [N] [ENTER]

IF DESIRED, ENTER A COMMENT LINE TO BE ADDED TO THE DATA
BLOCK DESCRIBING THE MATERIAL; TYPE [ENTER] FOR NONE.

>[30/-30/0/45/-45]5S from IM7/8551-7 hoop-wound at 0 deg from Z-axis.

DATA BLOCK COMPLETE. PRESS ANY KEY TO CONTINUE. [ENTER]

FILE GENERATION COMPLETE. PRESS ANY KEY TO CONTINUE. [ENTER]

Figure 12 illustrates the contents of the resulting ANSYS interface file. Note the control commands at the beginning and end of the file; these commands allow the materials data file to be used as an ANSYS User file. This file would be used independently, but in conjunction with a batch input file containing the finite element mesh data, loading data, and commands referencing the materials data file. In ANSYS, the X-axis corresponds to axial direction, the Y-axis corresponds to the radial direction, and the Z-axis is equivalent to the hoop or circumferential direction.

5 Current Status and Future Work

The computer-aided analysis system described in this paper, the CDI, is currently operational on a personal computer at the level of functionality described above. The system has been tested and documented [10]. In addition, the CDI is undergoing upward migration to a workstation environment. With enhancements and modifications, the CDI will be implemented on an Apollo platform running the Unix operating system. Testing and documentation of this system is also planned.

To achieve full functionality, the Laminate Design component must be expanded to include a fully three-dimensional classical laminate design model, and the Interface File Generation component must include three-dimensional transformation capabilities. Further system enhancements will include the addition of other laminate design models reflecting the particular characteristics and behaviors of thick composite laminates and the ability to generate interface files for general

three-dimensional (non-axisymmetric) structures. The modularity currently inherent in the CDI system will effectively support such enhancement and expansion.

Additionally, a component that is designed to compare the results of the various laminate design models is proposed, using either special R:BASE report formats or the exportation of data into spreadsheet and/or graphics programs. Other possible enhancements might include the integration of additional finite element analysis program interfaces, and the incorporation of a graphics capability into both the Laminate Design and Interface File Generation components, allowing the user to visualize laminates as they are being defined and oriented on finite element models.

6 Summary

This paper outlined an approach for merging database technology with finite element analysis applications in the domain of composite material properties. One scenario for the overall process of analysis and design using fiber-reinforced composite materials was reviewed and the complexity of the required integration process was emphasized. A computer-aided system was proposed to automate design and analysis using composite materials. An experimental implementation of the system, the Composites Database Interface (CDI), was presented and its contributions to overcoming the problems described above and enhancing the integration process were investigated.

The overall architecture of the CDI system using existing DBMS and FEA technologies was described. The relationship between the CDI and external software components was explained in terms of the functionality provided by each component and its data requirements. The paper then discussed system integration, which was a key project objective, and how the database management system provided an essential role in achieving it. The integration process was described in detail, and finally, the current status of the prototype was indicated and future enhancements were suggested.

References

- [1] Microrim, Inc., *R:BASE For DOS*, Version 2.1, Redmond, WA (1988).
- [2] Tsai, S.W., *Composites Design*, Think Composites, Dayton, OH (1987).
- [3] Halpin, J.C., *Primer on Composite Materials: Analysis*, Technomic, Lancaster, PA (1984).
- [4] Tretheway, B.R., Wilkins, D.J., and Gillespie, J.W., *Three-Dimensional Elastic Properties of Laminated Composites*, Report No. 89-04, Center for Composite Materials, University of Delaware, Newark, DE (June, 1989).
- [5] Hoskin, B.C. and Baker, A.A., Editors, *Composite Materials for Aircraft Structures*, AIAA Education Series, American Institute of Aeronautics and Astronautics, New York, NY (1986).
- [6] Swanson Computer Systems, Inc., *ANSYS*, Revision 4.4, Houston, PA (1989).
- [7] Hallquist, J.O., *MAZE—An Input Generator for DYNA2D and NIKE2D*, Report UCID-19029, Revision 2, Lawrence Livermore National Laboratory, Livermore, CA (June, 1983).

- [8] Boeing Computer Services, *RIM*, Version 7.0, Seattle, WA (1985).
- [9] Rasdorf, W.J., "Database Management System Extensions for Engineering Applications," *Computers in Mechanical Engineering (CIME)*, American Society of Mechanical Engineers, Volume 5, Number 5, Pages 62-69 (March, 1987).
- [10] Spainhour, L.K. and Rasdorf, W.J., *Composites Database Interface*, Users Manual, Ballistic Research Laboratory, Aberdeen Proving Grounds, MD (January, 1990).
- [11] Thompson, D.M. and Griffin, O.H. Jr., "2D to 3D Global/Local FEA of Composite Laminates," *Proceedings of the American Society for Composites* (October, 1989).

Appendix: Database Schema

The appendix contains the database schema for the experimental prototype of the Composites Database, as implemented in the R:BASE database management system. Each table is documented and the associated columns are enumerated. A brief description is provided for each table and column.

- UNICOMP (Unid, Uniname, Class, Fiber, Matrix, Volfrac, Prodform, Process)

The UNICOMP relation contains identification and descriptive information concerning unidirectional ply composites.

- UNID: Unique database identifier of a unidirectional composite
- UNINAME: Commonly used composite name or identification
- CLASS: Composite class, typically same as fiber type
- FIBER: Name of fiber used in the composite
- MATRIX: Name of matrix used in the composite
- VOLFRAC: Volume fraction of fiber in the composite
- PRODFORM: Form in which the composite product is available
- PROCESS: Processing which the composite has undergone

- UNIPROP (Unid, E1, E2, Alp1, Alp2, Nu12, G12, Dens, Plythick)

The UNIPROP relation contains material property data about unidirectional ply composites, specifically that which is relevant to finite element analysis.

- UNID: Unique database identifier of a unidirectional composite
- E1: Modulus of elasticity in fiber direction, in psi
- E2: Modulus of elasticity transverse to fiber, in psi
- ALP1: Coefficient of thermal expansion in fiber direction, in in/in-°C
- ALP2: Coefficient of thermal expansion transverse to fiber, in in/in-°C
- NU12: Major Poisson's ratio, unitless quantity
- G12: Shear modulus of elasticity, in psi
- DENS: Composite density, in lbf-s²/in
- PLYTHICK: Thickness of a single composite ply, in inches

- LAMINATE (Lamid, Unid, Lamcode, Lamthick)

The LAMINATE relation contains the information necessary for complete identification of a laminated composite.

- LAMID: Unique database identifier of a laminated composite
- UNID: Unidirectional composite used in the laminate
- LAMCODE: Laminate code indicating ply orientations and thicknesses
- LAMTHICK: Thickness of the composite laminate, in inches

- LAMSTIFF (Lamid, Runid, Method, Assump, Ex, Ey, Ez, ExBend, EyBend, Nuxy, Nuyz, Nuxz, Gxy, Gyz, Gxz)

The LAMSTIFF relation contains material stiffness data about thick laminated composites necessary to perform axisymmetric finite element analysis. In addition, it contains the data required to uniquely identify a stiffness analysis run.

- LAMID: Unique database identifier of a laminated composite
- RUNID: Identifier of stiffness analysis run number
- METHOD: Method used for generating stiffness properties
- ASSUMP: Three-dimensional ply/laminate assumptions used in generating laminate stiffness properties (note field)
- EX: Major elastic modulus in X direction (at zero degrees rotation), in psi
- EY: Minor elastic modulus in Y direction (at ninety degrees rotation), in psi
- EZ: Elastic modulus in Z direction (through thickness), in psi
- EXBEND: Bending stiffness around X axis, in psi
- EYBEND: Bending stiffness around Y axis, in psi
- NUXY: Major Poisson's ratio for the laminate in X-Y plane, unitless quantity
- NUYZ: Poisson's ratio for the laminate in Y-Z plane, unitless quantity
- NUXZ: Poisson's ratio for the laminate in X-Z plane, unitless quantity
- GXY: Shear modulus of elasticity in X-Y plane, in psi
- GYZ: Shear modulus of elasticity in Y-Z plane, in psi
- GXZ: Shear modulus of elasticity in X-Z plane, in psi

- LAMTHERM (Lamid, Alpx, Alpy, Alpxy, Alpz)

The LAMTHERM relation contains thermal expansion data about laminated composites necessary to perform finite element analysis.

- LAMID: Unique database identifier of a laminated composite
- ALPX: Major coefficient of thermal expansion in X direction, in in/in-°C
- ALPY: Minor coefficient of thermal expansion in Y direction, in in/in-°C
- ALPXY: Coefficient of thermal expansion at forty-five degrees in X-Y plane, in in/in-°C
- ALPZ: Coefficient of thermal expansion in Z direction, in in/in-°C

Figure 1: Composites Database Interface System Architecture

Figure 2: Data Flow for Data Integration With ANSYS

Figure 3: Large Tank to be Constructed of Thick Laminated Composites

Figure 4: Finite Element Model of Tank Showing Laminate Orientation

Figure 5: CDI Main Menu

*** DATABASE ACTIONS (Select an action) ***

- (1) Enter database data
- (2) Modify database data
- (3) Enter R:BASE directly
- (4) Exit to previous menu

Figure 6: Database Actions Menu

*** ENTER DATABASE DATA (Select a data category) ***

- (1) Unidirectional composites
- (2) Laminated composites
- (3) Exit to previous menu

Figure 7: Enter Database Data Menu

UNIDIRECTIONAL COMPOSITE MATERIALS

IDENTIFICATION CODE: U1

COMPOSITE NAME: IM7/8551-7

TYPE OR CLASS: CFRP

FIBER: IM7

MATRIX: 8551-7

FORM IN WHICH THE
PRODUCT IS AVAILABLE: Tape or cloth

PROCESSING WHICH THE
COMPOSITE HAS UNDERGONE: None

Figure 8: Data Entry Form for Ply Identification Data

UNIDIRECTIONAL COMPOSITE MATERIALS	
IDENTIFICATION CODE: U1 COMPOSITE NAME: IM7/8551-7 CLASS: CFRP	FIBER: IM7 MATRIX: 8551-7
ELASTIC MODULI: EX: <input type="text" value="23.0E6"/> (psi) EY: <input type="text" value="1.21E6"/> (psi)	THERMAL EXPANSION COEF: ALPX: <input type="text" value=".24E-6"/> (in/in- C) ALPY: <input type="text" value="14E-6"/> (in/in- C)
POISSON'S RATIOS: NUXY(MAJOR): <input type="text" value="0.34"/> NUYX(MINOR): <input type="text" value="0.018"/>	SHEAR MODULUS OF ELASTICITY: GXY: <input type="text" value="0.83E6"/> (psi)
DENSITY: <input type="text" value="148E-6"/> (lbf-s ² /in ⁴ = lbf/in ³ / 386.4 in/s ²)	

Figure 9: Data Entry Form for Ply Material Property Data

*** LAMINATE DESIGN MENU (Make a selection) ***

(1) Classical laminated stiffness theory (Tsai)

(2) Three-dimensional stiffness theory (STIF3D)

(3) Classical thermal expansion theory

(4) Exit to previous menu

Figure 10: Laminate Design Menu

*** GENERATE INTERFACE FILES (Select analysis program) ***

ANSYS MAZE INGRID Exit

Figure 11: Generate Interface Files Menu


```

TANK1
C*** [30/-30/0/45/-45]5S from IM7/8551-7 hoop-wound at 0 deg
C*** from Z-axis
C***
C*** The following data block defines material properties for the
C*** above listed composite material
C***
*SET,AR10,ARG1
*IF,ARG2,GT,0,:LB1
*SET,ARG2,ARG1
:LB1
*IF,ARG3,GT,0,:LB2
*SET,ARG3,1
:LB2
MP,EX,AR10,1210000.
MP,EY,AR10,3059484.
MP,EZ,AR10,9361206.
MP,ALPX,AR10,0.000014
MP,ALPY,AR10,0.000009
MP,ALPZ,AR10,0.000004
MP,NUXY,AR10,0.450000
MP,NUYZ,AR10,0.928000
MP,NUXZ,AR10,0.099999
MP,DENS,AR10,0.000148
MP,GXY,AR10,4366850.
*IF,AR10,GE,ARG2,:LB3
*SET,AR10,AR10+ARG3
*GO,:LB2
:LB3
/EOF

```

Figure 12: ANSYS Material Data File for [30/-30/0/45/-45]5S Laminate and Axisymmetric Cylinder