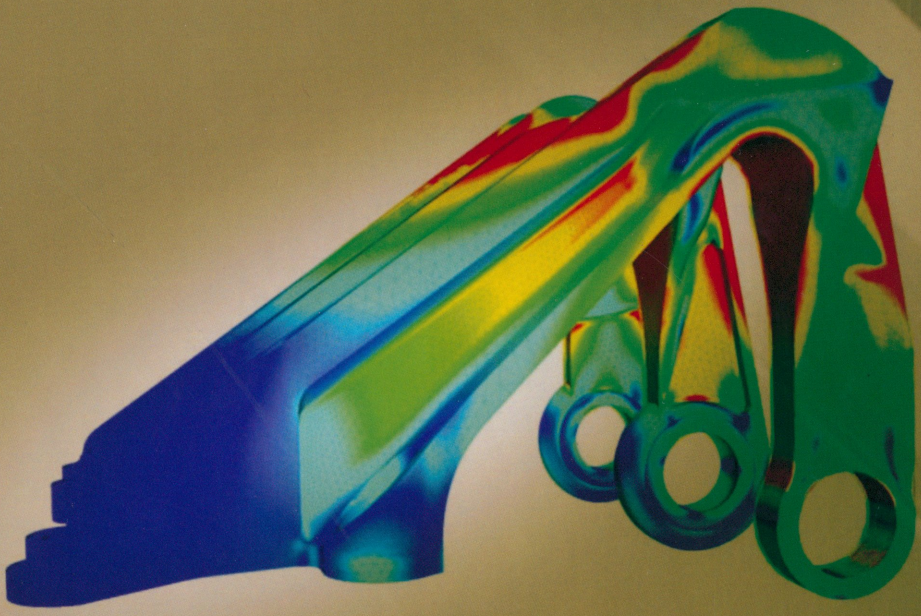


FINITE ELEMENTS IN ENGINEERING

SAUNDRA PELTIER

M.Hill Didactics Co.

Finite Elements in Engineering

Finite Elements in Engineering

Saundra Peltier



M.Hill Didactics Co.
www.mhilldidactics.com

Cataloging in Publication Data
Courtesy: M.Hill Didactics Co. < www.mhilldidactics.com>

Peltier, Saundra, author.
Finite Elements In Engineering / Saundra Peltier.
pages cm
Includes bibliographical references and index.
ISBN 9789353275921

1. Finite element method. 2. Engineering mathematics.

LCC: TA347.F5 2019| DDC 620.001/51 23

M.Hill Didactics Co.
102 suite, Victoria Street,
New Jersey , 7097
United States of America
enquiry@mhilldidactics.com

© 2019 by **M.Hill Didactics Co.**

All rights reserved. No part of this book may be reproduced, in any form or by any means, without permission in writing from the publisher. Company and product names mentioned herein are the trademarks or registered trademarks of their respective owners.

Finite Elements in Engineering
Saundra Peltier
ISBN: 978-9-35327-592-1

PRINTED IN INDIA

Contents

	Preface	**ix**
1.	**Introduction and Structural Analysis**	**1**
	1. Introduction	1
	2. Computer Programming	1
	3. Structural Analysis	8
	4. Case Study: Bending of a Tapered Beam	18
	Conclusion	19
	References	19
2.	**Continuum Mechanics Problems**	**23**
	1. Introduction	23
	2. Continuum Mechanics Equations	24
	3. Classification of Partial Differential Equations	30
	4. Methods for Solving Harmonic and Biharmonic Equations	40
	Conclusion	45
	References	45
3.	**Finite Element Analysis of Harmonic Problems**	**50**
	1. Introduction	50
	2. Derivation of the Element Stiffness Matrix	51
	3. Assembly of the Overall Stiffness Matrix	57
	4. Comparison with the Finite Difference Approach	59
	5. Variational Formulation	62
	6. Boundary Conditions	64
	7. Solution of the Linear Equations	69
	8. Convergence of Finite Element Methods	73
	9. A Computer Program for Harmonic Problems	78
	Conclusion	81
	References	82

4. Finite Element Meshes **86**

1. Introduction 86
2. Choice of Mesh 87
3. Mesh Data in Numerical Form 87
4. Generation of Mesh Data 90
5. Mesh Modification 102

Conclusion 104

References 105

5. Some Harmonic Problems **107**

1. Introduction 107
2. Case Study: Downstream Viscous Flow in a Rectangular Channel 108
3. Case Study: Torsion of Prismatic Bars 115

Conclusion 119

References 119

6. Finite Element Analysis of Biharmonic Problems **122**

1. Introduction 122
2. Derivation of the Element Stiffness Matrix 123
3. Assembly of the Overall Stiffness Matrix 125
4. Boundary Conditions 126
5. A Computer Program for Problems of the Biharmonic Plane Strain or Plane Stress Type 128

Conclusion 131

References 132

7. Some Biharmonic Problems **136**

1. Introduction 136
2. Case Study: Plane Strain Compression 136
3. Case Study: Stresses in Concentric Cylinders 138
4. Case Study: Stress Concentration near a Hole in a Flat Plate 143

Conclusion 153

References 154

8. Further Applications 157
1. Introduction 157
2. Axi-symmetric Problems 157
3. Higher-order Elements 163
4. Three-dimensional Problems 164
5. Biharmonic Problems Involving Incompressible Materials 165
6. Plate and Shell Problems 166
7. Isoparametric Elements 169
8. Nonlinear Problems 173
Conclusion 177
References 178

9. FEA of Elliptic Boundary Value Problems 181
1. Introduction 181
2. Galerkin Approximation 182
3. Existence and uniqueness of Galerkin Approximations 183
Conclusion 193
References 193

10. Variational Formulation of Boundary Value Problems 197
1. Introduction 197
2. Formulation of Variational Boundary-Value problem 198
3. Coercive Bilinear Forms 200
4. Existence and Uniqueness of Solution 202
Conclusion 211
References 211

11. Various Types of Finite Element Methods 214
1. Introduction 214
2. Stiffness matrix 216
Conclusion 230
References 231

Index 235

Preface

The finite element method approximates a structure as an assembly of elements or components with various forms of connection between them and each element of which has an associated stiffness. Thus, a continuous system such as a plate or shell is modeled as a discrete system with a finite number of elements interconnected at finite number of nodes and the overall stiffness is the result of the addition of the stiffness of the various elements. The application of finite difference method for engineering problems involves replacing the governing differential equations and the boundary condition by suitable algebraic equations.

The formulation for structural analysis is generally based on the three fundamental relations: equilibrium, constitutive and compatibility. There are two major approaches to the analysis: Analytical and Numerical. Analytical approach which leads to closed-form solutions is effective in case of simple geometry, boundary conditions, loadings and material properties. However, in reality, such simple cases may not arise. As a result, various numerical methods are evolved for solving such problems which are complex in nature. Engineers use finite element analysis software to reduce the number of physical prototypes and experiments and optimize components in their design phase to develop better products, faster while saving on expenses.

The book *"Finite Elements in Engineering"* provides an integrated approach to finite element methodologies. The development of finite element theory is combined with involving engineering applications. The steps used in the development of the theory are implemented in complete, self-contained computer programs. It is compiled in such a manner, that it will provide in-depth knowledge about the theory and practice of the subject. Coherent flow of topics, student-friendly Language and extensive use of example make this book an invaluable source of knowledge. This book is ideal for both undergraduate and postgraduate students and also as a learning resource to practicing engineers.

- Saundra Peltier

1 Introduction and Structural Analysis

OBJECTIVES

After studying this chapter, you will be able to know:

- Computer Programming
- Structural Analysis
- Bending of a Tapered Beam

1. INTRODUCTION

The primary role of structural study and analysis is to determine the internal actions and the support reactions of a structure subjected to mechanical loads, imposed deformations and settlements of supports. Structures are classified into two broad categories: statically determinate structures and statically indeterminate structures.

A structure, as it relates to civil engineering, is a system of interconnected members used to support external loads. Structural analysis is the prediction of the response of structures to specified arbitrary external loads. During the preliminary structural design stage, a structure's potential external load is estimated, and the sizes of the structure's interconnected members are determined based on the estimated loads. Structural analysis establishes the relationship between a structural member's expected external load and the structure's corresponding developed internal stresses and displacements that occur within the member when in service. This is necessary to ensure that the structural members satisfy the safety and the serviceability requirements of the local building code and specifications of the area where the structure is located.

Structural analysis is the determination of the effects of loads on physical structures and their components. Structures subject to this type of analysis include all that must withstand loads, such as buildings, bridges, aircraft and ships. Structural analysis employs the fields of applied mechanics, materials science and applied mathematics to compute a structure's deformations, internal forces, stresses, support reactions, accelerations, and stability. The results of the analysis are used to verify a structure's fitness for use, often precluding physical tests. Structural analysis is thus a key part of the engineering design of structures.

2. COMPUTER PROGRAMMING

Computer programming is the process of designing and building an executable computer program to accomplish a specific computing result or to

perform a specific task. Programming involves tasks such as: analysis, generating algorithms, profiling algorithms' accuracy and resource consumption, and the implementation of algorithms in a chosen programming language (commonly referred to as coding). The source code of a program is written in one or more languages that are intelligible to programmers, rather than machine code, which is directly executed by the central processing unit. The purpose of programming is to find a sequence of instructions that will automate the performance of a task (which can be as complex as an operating system) on a computer, often for solving a given problem. Proficient programming thus often requires expertise in several different subjects, including knowledge of the application domain, specialized algorithms, and formal logic.

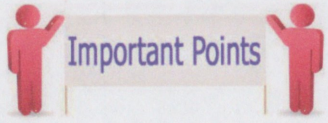
Tasks accompanying and related to programming include: testing, debugging, source code maintenance, implementation of build systems, and management of derived artifacts, such as the machine code of computer programs. These might be considered part of the programming process, but often the term software development is used for this larger process with the term programming, implementation, or coding reserved for the actual writing of code. Software engineering combines engineering techniques with software development practices. Reverse engineering is a related process used by designers, analysts and programmers to understand and re-create/re-implement.

Modern Programming

Quality requirements

Whatever the approach to development may be, the final program must satisfy some fundamental properties. The following properties are among the most important:

- **Reliability:** how often the results of a program are correct. This depends on conceptual correctness of algorithms, and minimization of programming mistakes, such as mistakes in resource management (e.g., buffer overflows and race conditions) and logic errors (such as division by zero or off-by-one errors).
- **Robustness**: how well a program anticipates problems due to errors (not bugs). This includes situations such as incorrect, inappropriate or corrupt data, unavailability of needed resources such as memory, operating system services, and network connections, user error, and unexpected power outages.



The source code of a program is written in one or more languages that are intelligible to programmers, rather than machine code, which is directly executed by the central processing unit.

- **Usability:** the ergonomics of a program: the ease with which a person can use the program for its intended purpose or in some cases even unanticipated purposes. Such issues can make or break its success even regardless of other issues. This involves a wide range of textual, graphical, and sometimes hardware elements that improve the clarity, intuitiveness, cohesiveness and completeness of a program's user interface.
- **Portability:** the range of computer hardware and operating system platforms on which the source code of a program can be compiled/interpreted and run. This depends on differences in the programming facilities provided by the different platforms, including hardware and operating system resources, expected behavior of the hardware and operating system, and availability of platform-specific compilers (and sometimes libraries) for the language of the source code.
- **Maintainability:** the ease with which a program can be modified by its present or future developers in order to make improvements or customizations, fix bugs and security holes, or adapt it to new environments. Good practices during initial development make the difference in this regard. This quality may not be directly apparent to the end user but it can significantly affect the fate of a program over the long term.
- **Efficiency/Performance:** Measure of system resources a program consumes (processor time, memory space, slow devices such as disks, network bandwidth and to some extent even user interaction): the less, the better. This also includes careful management of resources, for example cleaning up temporary files and eliminating memory leaks. This is often discussed under the shadow of a chosen programming language. Although the language certainly affects performance, even slower languages, such as Python, can execute programs instantly from a human perspective. Speed, resource usage, and performance are important for programs that bottleneck the system, but efficient use of programmer time is also important and is related to cost: more hardware may be cheaper.

Readability of Source Code

In computer programming, readability refers to the ease with which a human reader can comprehend the purpose, control flow, and operation of source code. It affects the aspects of quality above, including portability, usability and most importantly maintainability.

Readability is important because programmers spend the majority of their time reading, trying to understand and modifying existing source code, rather than writing new source code. Unreadable code often leads to

bugs, inefficiencies, and duplicated code. A study found that a few simple readability transformations made code shorter and drastically reduced the time to understand it.

Following a consistent programming style often helps readability. However, readability is more than just programming style. Many factors, having little or nothing to do with the ability of the computer to efficiently compile and execute the code, contribute to readability. Some of these factors include:

- Different indent styles (whitespace)
- Comments
- Decomposition
- Naming conventions for objects (such as variables, classes, procedures, etc.)

The presentation aspects of this (such as indents, line breaks, color highlighting, and so on) are often handled by the source code editor, but the content aspects reflect the programmer's talent and skills. Various visual programming languages have also been developed with the intent to resolve readability concerns by adopting non-traditional approaches to code structure and display. Integrated development environments (IDEs) aim to integrate all such help. Techniques like Code refactoring can enhance readability.

Algorithmic Complexity

The academic field and the engineering practice of computer programming are both largely concerned with discovering and implementing the most efficient algorithms for a given class of problem. For this purpose, algorithms are classified into orders using so-called Big O notation, which expresses resource use, such as execution time or memory consumption, in terms of the size of an input. Expert programmers are familiar with a variety of well-established algorithms and their respective complexities and use this knowledge to choose algorithms that are best suited to the circumstances.

Chess Algorithms as an Example

"Programming a Computer for Playing Chess" was a 1950 paper that evaluated a "minimax" algorithm that is part of the history of algorithmic complexity; a course on IBM's Deep Blue (chess computer) is part of the computer science curriculum at Stanford University.



Readability is important because programmers spend the majority of their time reading, trying to understand and modifying existing source code, rather than writing new source code.

Methodologies

The first step in most formal software development processes is requirements analysis, followed by testing to determine value modeling, implementation, and failure elimination (debugging). There exist a lot of differing approaches for each of those tasks. One approach popular for requirements analysis is Use Case analysis. Many programmers use forms of Agile software development where the various stages of formal software development are more integrated together into short cycles that take a few weeks rather than years. There are many approaches to the Software development process.

Popular modeling techniques include Object-Oriented Analysis and Design (OOAD and Model-Driven Architecture (MDA). The Unified Modeling Language (UML) is a notation used for both the OOAD and MDA.

A similar technique used for database design is Entity-Relationship Modeling (ER Modeling). Implementation techniques include imperative languages (object-oriented or procedural), functional languages, and logic languages.

Measuring Language Usage

It is very difficult to determine what are the most popular modern programming languages. Methods of measuring programming language popularity include: counting the number of job advertisements that mention the language, the number of books sold and courses teaching the language (this overestimates the importance of newer languages), and estimates of the number of existing lines of code written in the language (this underestimates the number of users of business languages such as COBOL).

Some languages are very popular for particular kinds of applications, while some languages are regularly used to write many different kinds of applications. For example, COBOL is still strong in corporate data centers often on large mainframe computers, Fortran in engineering applications, scripting languages in Web development, and C in embedded software. Many applications use a mix of several languages in their construction and use. New languages are generally designed around the syntax of a prior language with new functionality added, (for example C++ adds object-orientation to C, and Java adds memory management and bytecode to C++, but as a result, loses efficiency and the ability for low-level manipulation).

Debugging

Debugging is a very important task in the software development process since having defects in a program can have significant consequences for its users. Some languages are more prone to some kinds of faults because their

specification does not require compilers to perform as much checking as other languages. Use of a static code analysis tool can help detect some possible problems. Normally the first step in debugging is to attempt to reproduce the problem. This can be a non-trivial task, for example as with parallel processes or some unusual software bugs. Also, specific user environment and usage history can make it difficult to reproduce the problem.

After the bug is reproduced, the input of the program may need to be simplified to make it easier to debug. For example, when a bug in a compiler can make it crash when parsing some large source file, a simplification of the test case that results in only few lines from the original source file can be sufficient to reproduce the same crash. Trial-and-error/divide-and-conquer is needed: the programmer will try to remove some parts of the original test case and check if the problem still exists. When debugging the problem in a GUI, the programmer can try to skip some user interaction from the original problem description and check if remaining actions are sufficient for bugs to appear. Scripting and breakpointing is also part of this process.

Debugging is often done with IDEs like Eclipse, Visual Studio, Xcode, Kdevelop, NetBeans and Code: Blocks. Standalone debuggers like GDB are also used, and these often provide less of a visual environment, usually using a command line. Some text editors such as Emacs allow GDB to be invoked through them, to provide a visual environment.

- **Coding the Program:** As the programmer, your next step is to code the program-that is, to express your solution in a programming language. You will translate the logic from the flowchart or pseudocode-or some other tool-to a programming language. As we have already noted, a programming language is a set of rules that provides a way of instructing the computer what operations to perform. There are many programming languages: BASIC, COBOL, Pascal, FORTRAN, and C are some examples. You may find yourself working with one or more of these. We will discuss the different types of languages in detail later in this chapter.
- Although, programming languages operate grammatically, somewhat like the English language, they are much more precise. To get your program to work, you have to follow exactly the rules-the syntax-of the language you are using. Of course, using the language correctly is no guarantee that your program will work, any more than speaking grammatically correct English means you know what you are talking about. The point is that correct use of the language is the required first step. Then your coded program must be keyed, probably using a terminal or personal computer, in a form the computer can understand.

One more note here: Programmers usually use a text editor, which is somewhat like a word processing program, to create a file that contains the program. However, as a beginner, you will probably want to write your program code on paper first.

- **Testing the Program:** Some experts insist that a well-designed program can be written correctly the first time. In fact, they assert that there are mathematical ways to prove that a program is correct. However, the imperfections of the world are still with us, so most programmers get used to the idea that their newly written programs probably have a few errors. This is a bit discouraging at first, since programmers tend to be precise, careful, detail-oriented people who take pride in their work. Still, there are many opportunities to introduce mistakes into programs, and you, just as those who have gone before you, will probably find several of them.

  Eventually, after coding the program, you must prepare to test it on the computer. This step involves these phases:

- **Desk-checking:** This phase, similar to proofreading, is sometimes avoided by the programmer who is looking for a shortcut and is eager to run the program on the computer once it is written. However, with careful desk-checking you may discover several errors and possibly save yourself time in the long run. In desk-checking you simply sit down and mentally trace, or check, the logic of the program to attempt to ensure that it is error-free and workable. Many organizations take this phase a step further with a walkthrough, a process in which a group of programmers-your peers-review your program and offer suggestions in a collegial way.

- **Translating:** A translator is a program that (1) checks the syntax of your program to make sure the programming language was used correctly, giving you all the syntax-error messages, called diagnostics, and (2) then translates your program into a form the computer can understand. A by-product of the process is that the translator tells you if you have improperly used the programming language in some way. These types of mistakes are called syntax errors. The translator produces descriptive error messages. For instance, if in FORTRAN you mistakenly write N=2 *(I+J))-which has two closing parentheses instead of one-you will get a message that says, "UNMATCHED PARENTHESES." (Different translators may provide different wording for error messages.) Programs are most commonly translated by a compiler. A compiler translates your entire program at one time. The translation involves your original program, called a source module, which is transformed by a compiler into an object module. Prewritten programs from a system library may be added during the link/load phase, which results in a load module. The load module can then be executed by the computer.

- **Debugging:** A term used extensively in programming, debugging means detecting, locating, and correcting bugs (mistakes), usually by running the program. These bugs are logic errors, such as telling a computer to repeat an operation but not telling it how to stop repeating. In this phase you run the program using test data that you devise. You must plan the test data carefully to make sure you test every part of the program.

3. STRUCTURAL ANALYSIS

Structural Analysis is the branch which involves in the determination of behaviour of structures such as Buildings, Bridges, Dams, Towers, Offshore Platforms, Retaining Walls, Truss, Foundations and Stadiums. The concepts of Statics provide basis for solutions to the structures. Computer software are also being used for the calculation of forces, bending moment, stress, strain and deformation or deflection for complex structural systems. Sophisticated equipments are also used for investigating the structural response under different types of loading. Basic concepts of applied mechanics are essential before structural analysis. Structural analysis provides the basis for Structural Design. A structure can be statically determinate or indeterminate.

Scope of Structural Analysis

The structural analysis process can be broadly classified into three main categories:

Static analysis determines the internal forces and displacements due to time-independent loading conditions.

Stability analysis deals with products that are subject to compressed time-independent forces.

Vibration analysis determines the natural frequencies/eigenvalues and corresponding mode shapes (eignenfunctions) of vibration in the product.

Structural analysis of any product is based on the following assumptions.

1. The product material is completely elastic
2. Superposition principle - any factor caused by different loads that act simultaneously are equal to the algebraic or geometric sum of this factor due to each load separately
3. The deformation of the product caused by the applied loads do not change the original design diagram



Computer software is also being used for the calculation of forces, bending moment, stress, strain and deformation or deflection for complex structural systems.

4. Relationship between stress and strain is linearEarlier, structural analysis was a tedious chore that involved solving mathematical equations manually. The introduction of CAE software has made structural analysis relatively simple, though it still takes a highly skilled engineer to properly interpret the results.

Structural Analysis – the Process

All structural design services companies start by simulating a geometric model of the product. This model needs to be meshable into a correct finite element mesh. This is done in order to ensure that the CAD geometry will mesh and will provide important data like stresses, displacements or temperature distribution with accepted accuracy. There are three steps involved in this process:

Defeaturing: this step involves removing unwanted geometry features from the CAD model. A few examples of such features are chamfers, holes, outside fillets, etc.

Idealization: This basically means simplifying the CAD model to reduce the analysis time without compromising on the precision of the computed results.

Clean-up: Clean-up is done to ensure that the model geometry satisfies high quality requirements to be mesahble. The structural analysis service team uses CAD quality control tools for this.

Once you create a geometric model of the product, the mathematical model can be split using the process of discretization. The geometry, loads and restraints are all discretized. This process is called as meshing. Simply put, meshing is a discredtized representation of the product's geometry.

Structural Analysis Services

Structural analysis of a product is an iterative process. The structural analyst creates a discrete geometric model based on the designer's inputs and by using some CAE software tools. The designer then validates the products based on the simulation results. After optimization, the product design is again sent to the structural analyst, and the cycle continues. While a few companies have in-house CAD/CAE designers, they are not in a position to purchase the latest CAE software required for the structural analysis. This is where they outsource the structural analysis process to companies that have this expertise.

Statically Determinate Structures

A structure is termed as statically determinate if all of its unknown forces can be determined by using only the equations of static equilibrium. For example cantilever, simply supported beam etc.

Structures and Loads

A structure refers to a body or system of connected parts used to support a load. Important examples related to Civil Engineering include buildings, bridges, and towers; and in other branches of engineering, ship and aircraft frames, tanks, pressure vessels, mechanical systems, and electrical supporting structures are important. To design a structure, an engineer must account for its safety, aesthetics, and serviceability, while considering economic and environmental constraints. Other branches of engineering work on a wide variety of non-building structures.

Classification of Structures

A structural system is the combination of structural elements and their materials. It is important for a structural engineer to be able to classify a structure by either its form or its function, by recognizing the various elements composing that structure. The structural elements guiding the systemic forces through the materials are not only such as a connecting rod, a truss, a beam, or a column, but also a cable, an arch, a cavity or channel, and even an angle, a surface structure, or a frame.

Loads

One important element of the structural analysis is the accurate analysis of the estimation of structural loads to be endured. Structural loads are the forces on a part of a structure, or on the complete structure, and evaluation of their consequences is conducted by structural analysis. Overload loading may result into failure of structures, and such conditions should be considered during the design of the structures. For buildings and bridges, the main vertical loads are gravity loads, including the structure weight and the permanent accessories, known as dead loads. The live loads are the concentrated loads, or distributed loads over large areas such as floors. The horizontal loading on buildings is due to wind and inertial forces caused by earthquakes.

Once the dimensional requirement for a structure have been defined, it becomes necessary to determine the loads the structure must support. Structural design, therefore begins with specifying loads that act on the structure. The design loading for a structure is often specified in building codes. There are two types of codes: general building codes and design codes, engineers must satisfy all of the code's requirements in order for the structure to remain reliable.

There are two types of loads that structure engineering must encounter in the design. The first type of loads are dead loads that consist of the weights of the various structural members and the weights of any objects that are permanently attached to the structure. For example, columns, beams, girders,

the floor slab, roofing, walls, windows, plumbing, electrical fixtures, and other miscellaneous attachments. The second types of loads are live loads which vary in their magnitude and location. There are many different types of live loads like building loads, highway bridge loads, railroad bridge loads, impact loads, wind loads, snow loads, earthquake loads, and other natural loads.

Modes of Structural Analysis

Structural analysis is carried out by an examination of the real structure, on a model of the structure created on some scale, and by the utilization of mathematical models. Tests are conducted on the real structure when production is required of similar structures in large quantities, like frames of a particular car, or when the test expenses are acceptable due to the significance of the task. When elements of the main structures are to be examined, then models are used for the estimation of the different loads to be endured. Most structural analyses are conducted on the mathematical models, in which the model could be elastic or inelastic, forces may be static or dynamic, and the model of the structure might be two dimensional or three dimensional.

Strength of Materials

There are several methods to determine the stresses in different structural members, such as columns and beams that may cause failure under loading. To conduct an accurate structural analysis, complete data concerning the structural loads, properties of materials, and stresses involved is essential. The approach related to the strength of materials is utilized for simple structural elements subjected to plain loads, such as shafts subjected to torsion, and axial loading of bars. The strength of a material is dependent upon its microstructure that can be changed by the application of various engineering techniques on the material, such as work hardening and grain boundary hardening. However, various mechanical characteristics of a material may deteriorate in an effort to increase the strength, such as when the grain size is decreased, the yield strength is increased but the material becomes brittle.

Methods Using Numerical Approximation

It is common practice to use approximate solutions of differential equations as the basis for structural analysis. This is usually done using numerical approximation techniques. The most commonly used numerical approximation in structural analysis is the Finite Element Method.

The finite element method approximates a structure as an assembly of elements or components with various forms of connection between them and each element of which has an associated stiffness. Thus, a continuous system such as a plate or shell is modeled as a discrete system with a finite number of elements interconnected at finite number of nodes and the overall stiffness is the result of

the addition of the stiffness of the various elements. The behaviour of individual elements is characterized by the element's stiffness (or flexibility) relation. The assemblage of the various stiffness's into a master stiffness matrix that represents the entire structure leads to the system's stiffness or flexibility relation. To establish the stiffness (or flexibility) of a particular element, we can use the mechanics of materials approach for simple one-dimensional bar elements, and the elasticity approach for more complex two- and three-dimensional elements. The analytical and computational development are best effected throughout by means of matrix algebra, solving partial differential equations.

Early applications of matrix methods were applied to articulated frameworks with truss, beam and column elements; later and more advanced matrix methods, referred to as "finite element analysis", model an entire structure with one-, two-, and three-dimensional elements and can be used for articulated systems together with continuous systems such as a pressure vessel, plates, shells, and three-dimensional solids. Commercial computer software for structural analysis typically uses matrix finite-element analysis, which can be further classified into two main approaches: the displacement or stiffness method and the force or flexibility method. The stiffness method is the most popular by far thanks to its ease of implementation as well as of formulation for advanced applications. The finite-element technology is now sophisticated enough to handle just about any system as long as sufficient computing power is available. Its applicability includes, but is not limited to, linear and non-linear analysis, solid and fluid interactions, materials that are isotropic, orthotropic, or anisotropic, and external effects that are static, dynamic, and environmental factors. This, however, does not imply that the computed solution will automatically be reliable because much depends on the model and the reliability of the data input.

Elasticity Methods

Elasticity methods are available generally for an elastic solid of any shape. Individual members such as beams, columns, shafts, plates and shells may be modeled. The solutions are derived from the equations of linear elasticity. The equations of elasticity are a system of 15 partial differential equations. Due to the nature of the mathematics involved, analytical solutions may only be produced for relatively simple geometries. For complex geometries, a numerical solution method such as the finite element method is necessary.

Analytical Methods

To perform an accurate analysis a structural engineer must determine information such as structural loads, geometry, support conditions, and material properties. The results of such an analysis typically include support reactions, stresses and displacements. This information is then compared to

criteria that indicate the conditions of failure. Advanced structural analysis may examine dynamic response, stability and non-linear behavior. There are three approaches to the analysis: the mechanics of materials approach (also known as strength of materials), the elasticity theory approach (which is actually a special case of the more general field of continuum mechanics), and the finite element approach. The first two make use of analytical formulations which apply mostly simple linear elastic models, leading to closed-form solutions, and can often be solved by hand. The finite element approach is actually a numerical method for solving differential equations generated by theories of mechanics such as elasticity theory and strength of materials. However, the finite-element method depends heavily on the processing power of computers and is more applicable to structures of arbitrary size and complexity.

Regardless of approach, the formulation is based on the same three fundamental relations: equilibrium, constitutive, and compatibility. The solutions are approximate when any of these relations are only approximately satisfied, or only an approximation of reality.

Limitations

Each method has noteworthy limitations. The method of mechanics of materials is limited to very simple structural elements under relatively simple loading conditions. The structural elements and loading conditions allowed, however, are sufficient to solve many useful engineering problems. The theory of elasticity allows the solution of structural elements of general geometry under general loading conditions, in principle. Analytical solution, however, is limited to relatively simple cases. The solution of elasticity problems also requires the solution of a system of partial differential equations, which is considerably more mathematically demanding than the solution of mechanics of materials problems, which require at most the solution of an ordinary differential equation.

The finite element method is perhaps the most restrictive and most useful at the same time. This method itself relies upon other structural theories (such as the other two discussed here) for equations to solve. It does, however, make it generally possible to solve these equations, even with highly complex geometry and loading conditions, with the restriction that there is always some numerical error. Effective and reliable use of this method requires a solid understanding of its limitations.

Strength of Materials Methods (Classical Methods)

The simplest of the three methods here discussed, the mechanics of materials method is available for simple structural members subject to specific loadings such as axially loaded bars, prismatic beams in a state of pure bending, and circular shafts subject to torsion. The solutions can under certain

conditions be superimposed using the superposition principle to analyze a member undergoing combined loading. Solutions for special cases exist for common structures such as thin-walled pressure vessels.

For the analysis of entire systems, this approach can be used in conjunction with statics, giving rise to the method of sections and method of joints for truss analysis, moment distribution method for small rigid frames, and portal frame and cantilever method for large rigid frames. Except for moment distribution, which came into use in the 1930s, these methods were developed in their current forms in the second half of the nineteenth century. They are still used for small structures and for preliminary design of large structures.

The solutions are based on linear isotropic infinitesimal elasticity and Euler–Bernoulli beam theory. In other words, they contain the assumptions (among others) that the materials in question are elastic, that stress is related linearly to strain, that the material (but not the structure) behaves identically regardless of direction of the applied load, that all deformations are small, and that beams are long relative to their depth. As with any simplifying assumption in engineering, the more the model strays from reality, the less useful (and more dangerous) the result.

Example

There are two commonly used methods to find the truss element forces, namely the Method of Joints and the Method of Sections. Below is an example that is solved using both of these methods. The first diagram below is the presented problem for which we need to find the truss element forces. The second diagram is the loading diagram and contains the reaction forces from the joints.

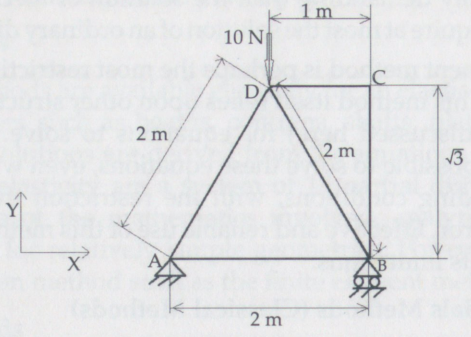


Fig. 1.1 Truss element forces.

IMPORTANT

The structural elements and loading conditions allowed, however, are sufficient to solve many useful engineering problems.

Since there is a pin joint at A, it will have two reaction forces. One in the x direction and the other in the y direction. At point B, we have a roller joint and hence we only have 1 reaction force in the y direction. Let us assume these forces to be in their respective positive directions (if they are not in the positive directions like we have assumed, then we will get a negative value for them).

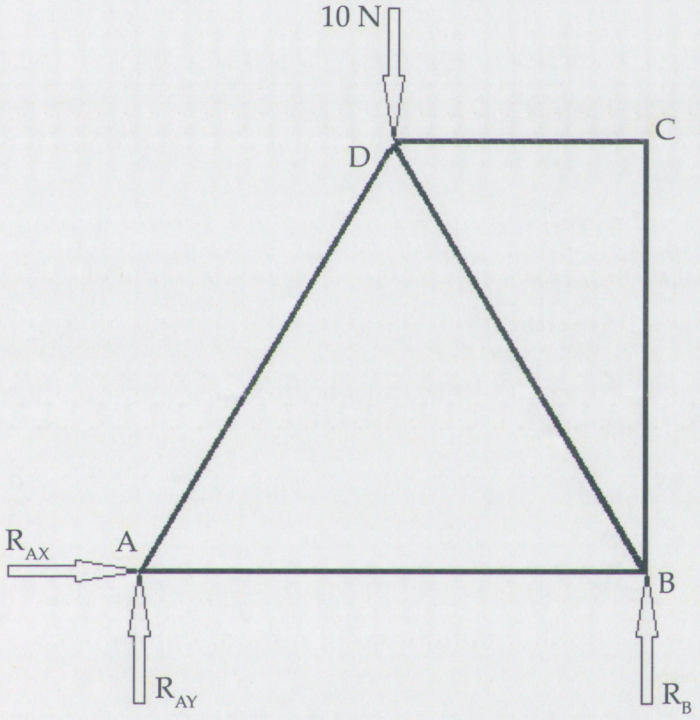


Fig. 1.2 Reaction forces from the joints.

Since the system is in static equilibrium, the sum of forces in any direction is zero and the sum of moments about any point is zero. Therefore, the magnitude and direction of the reaction forces can be calculated.

$$\sum M_A = 0 = -10*1 + 2*R_B \Rightarrow R_B = 5$$

$$\sum F_y = 0 = R_{Ay} + R_B - 10 \Rightarrow R_{Ay} = 5$$

$$\sum F_x = 0 = R_{Ax}$$

Method of Joints

This type of method uses the force balance in the x and y directions at each of the joints in the truss structure.

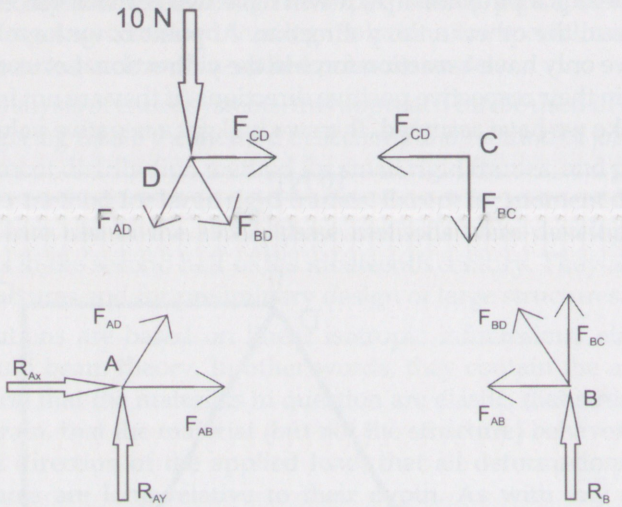


Fig. 1.3 Truss structure.

At A,

$$\sum F_y = 0 = R_{Ay} + F_{AD}\sin(60) = 5 + F_{AD}\frac{\sqrt{3}}{2} \Rightarrow F_{AD} = -\frac{10}{\sqrt{3}}$$

$$\sum F_x = 0 = R_{Ax} + F_{AD}\cos(60) + F_{AB} = 0 - \frac{10}{\sqrt{3}}\frac{1}{2} + F_{AB} \Rightarrow F_{AB} = \frac{5}{\sqrt{3}}$$

At D,

$$\sum F_y = 0 = -10 - F_{AD}\sin(60) - F_{BD}\sin(60) = -10 - \left(-\frac{10}{\sqrt{3}}\right)\frac{\sqrt{3}}{2} - F_{BD}\frac{\sqrt{3}}{2} \Rightarrow F_{BD} = -\frac{10}{\sqrt{3}}$$

$$\sum F_x = 0 = -F_{AD}\cos(60) + F_{BD}\cos(60) + F_{CD} = -\frac{10}{\sqrt{3}}\frac{1}{2} + \frac{10}{\sqrt{3}}\frac{1}{2} + F_{CD} \Rightarrow F_{CD} = 0$$

At C,

$$\sum F_y = 0 = -F_{BC} \Rightarrow F_{BC} = 0$$

Although we have found the forces in each of the truss elements, it is a good practice to verify the results by completing the remaining force balances.

$$\sum F_x = -F_{CD} = -0 = 0 \Rightarrow \text{verified}$$

At B,

$$\sum F_y = R_B + F_{BD}\sin(60) + F_{BC} = 5 + \left(-\frac{10}{\sqrt{3}}\right)\frac{\sqrt{3}}{2} + 0 = 0 \Rightarrow \text{verified}$$

$$\sum F_x = -F_{AB} - F_{BD}\cos(60) = \frac{5}{\sqrt{3}} - \frac{10}{\sqrt{3}}\frac{1}{2} = 0 \Rightarrow \text{verified}$$

Method of Sections

This method can be used when the truss element forces of only a few members are to be found. This method is used by introducing a single straight line cutting through the member whose force wants to be calculated. However this method has a limit in that the cutting line can pass through a maximum of only 3 members of the truss structure. This restriction is because this method uses the force balances in the x and y direction and the moment balance, which gives us a maximum of 3 equations to find a maximum of 3 unknown truss element forces through which this cut is made. Let us try to find the forces FAB, FBD and FCD in the above example

Method 1: Ignore the Right Side

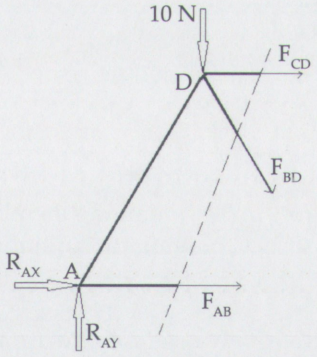


Fig. 1.4 Ignore the right side.

$$\sum M_D = 0 = -5*1 + \sqrt{3}*F_{AB} \Rightarrow F_{AB} = \frac{5}{\sqrt{3}}$$

$$\sum F_y = 0 = R_{Ay} - F_{BD}\sin(60) - 10 = 5 - F_{BD}\frac{\sqrt{3}}{2} - 10 \Rightarrow F_{BD} = -\frac{10}{\sqrt{3}}$$

$$\sum F_x = 0 = F_{AB} + F_{BD}\cos(60) + F_{CD} = \frac{5}{\sqrt{3}} - \frac{10}{\sqrt{3}}\frac{1}{2} + F_{CD} \Rightarrow F_{CD} = 0$$

Method 2: Ignore the Left Side

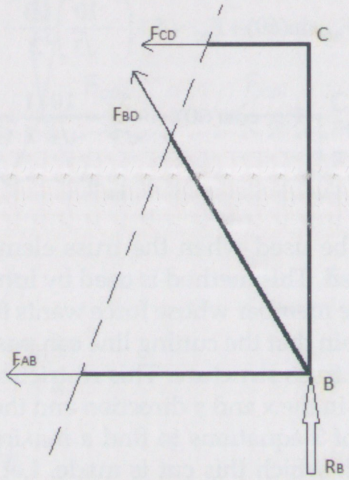


Fig. 1.5 Ignore the left side

$$\sum M_B = 0 = \sqrt{3} * F_{CD} \Rightarrow F_{CD} = 0$$

$$\sum F_y = 0 = F_{BD} \sin(60) + R_B = F_{BD} \frac{\sqrt{3}}{2} + 5 \Rightarrow F_{BD} = -\frac{10}{\sqrt{3}}$$

$$\sum F_x = 0 = -F_{AB} - F_{BD} \cos(60) - F_{CD} = -F_{AB} - \left(-\frac{10}{\sqrt{3}}\right)\frac{1}{2} - 0 \Rightarrow F_{AB} = \frac{5}{\sqrt{3}}$$

The truss elements forces in the remaining members can be found by using the above method with a section passing through the remaining members.

4. CASE STUDY: BENDING OF A TAPERED BEAM

Tapered beams are extensively used for structural applications due to their high stiffness-to-mass ratio. They provide many advantages over prismatic beams such as better shear carrying capacity, higher lateral stability, and weight savings. As it is known, axial stress $\sigma x = -My/t$, from Navier's flexure formula, may be used to estimate bending stresses in tapered beams to some extent, and this can be useful for primary design purposes. However, since the section modulus may vary along the axis of tapered beams, due to the additionally generated shear stress field, the maximum stress cannot necessarily occur at the cross section of the tapered beams where the largest bending moment is present. Nevertheless, classical beam theories do not predict the shear stress distributions in tapered beams if the taper angle is greater than 15°.

This study aims at combining the advanced mechanics of a material approach with the theory of elasticity for three different loading conditions applied at the free end of the circular cantilever tapered beams. Derived equations provide the stress distribution across the circular cantilever tapered beams subjected to axial tensile stress, bending moment, and transverse shear force. In order to verify the analytical calculations, a FEM model is employed, and its results show a reasonable agreement with the analytical results.

CONCLUSION

Structural analysis is a science that ensures that the structures are safe and fulfill the functions for which they were built. Safety requirements must be met so that a structure is able to serve its purpose with the minimum of costs. Structural concepts arise from the work of engineers from different fields with a common aim that the structure is functional, aesthetic, and economic.

Detailed planning of the structure usually comes from several studies made by town planners, investors, users, architects, and other engineers. In general, an architect is responsible to the investor and a structural engineer works in collaboration with the architect as an equal partner in a project. In some structures, such as industrial halls, bridges, and sports halls, a structural engineer has the main influence on the overall structural design and an architect is involved in aesthetic details. The process of analysis has to be repeated until the structure as a whole is optimal from all points of view, followed by final analysis and dimensioning.

REFERENCES

- Bebbington, Shaun (2014). What is coding. Tumblr. Archived from the original on April 29, 2020. Retrieved March 3, 2014.
- Bergstein, Brian (March 20, 2007). Fortran creator John Backus dies. NBC News. Archived from the original on April 29, 2020. Retrieved April 25, 2010.
- Ceruzzi, Paul E. (1998). History of Computing. Cambridge, Massachusetts: MIT Press. ISBN 9780262032551 – via EBSCOhost.
- Computer Authority to Speak Here. The Times. April 9, 1972. Retrieved October 13, 2018 – via Newspapers.com.
- da Cruz, Frank (March 10, 2020). Columbia University Computing History – Herman Hollerith. Columbia University. Columbia.edu. Archived from the original on April 29, 2020. Retrieved April 25, 2010.
- Dooley, John F. (2013). A Brief History of Cryptology and Cryptographic Algorithms. Springer Science & Business Media. pp. 12–3. ISBN 9783319016283.
- Eilam, Eldad (2005). Reversing: secrets of reverseengineering. John Wiley & Sons. ISBN 978-0-7645-7481-8.
- Elshoff, James L.; Marcotty, Michael (1982). Improving computer program readability to aid modification. Communications of the ACM. 25 (8): 512–521. doi:10.1145/358589.358596. S2CID 30026641.

- Enticknap, Nicholas (September 11, 2007). SSL/Computer Weekly IT salary survey: finance boom drives IT job growth.
- Evans, Claire L. (2018). Broad Band: The Untold Story of the Women Who Made the Internet. New York: Portfolio/Penguin. ISBN 9780735211759.
- Fowler, Charles B. (October 1967). The Museum of Music: A History of Mechanical Instruments. Music Educators Journal. 54 (2): 45–49. doi:10.2307/3391092. JSTOR 3391092. S2CID 190524140.
- Frances Holberton, Pioneer in Computer Languages, Dies. The Courier-Journal. December 12, 2001. Retrieved October 13, 2018 – via Newspapers.com.
- Mitchell, Robert (May 21, 2012). The Cobol Brain Drain. Computer World. Retrieved May 9, 2015.
- NIST To Develop Cloud Roadmap. InformationWeek. November 5, 2010. Computing initiative seeks to remove barriers to cloud adoption in security, interoperability, portability and reliability.
- Noel Sharkey (2007), A 13th Century Programmable Robot, University of Sheffield
- Photograph courtesy Naval Surface Warfare Center, Dahlgren, Virginia, from National Geographic Sept. 1947

MCQs

1. **Which of the following methods of Structural analysis is a Force method?**
   a. Slope deflection method
   b. Column analogy method
   c. Moment distribution method
   d. None of the above
2. **Which of the following methods of Structural analysis is a Displacement method?**
   a. Moment distribution method
   b. Column analogy method
   c. Three moment equation
   d. None of the above
3. **Programmer who writes System Software is called**
   a. System programmer
   b. Analysis programmer
   c. Train programmer
   d. Design programmer

4. **Programmer who works with high level languages and have better understanding with Applications are considered as**
   a. Design programmer
   b. Application programmer
   c. Analysis programmer
   d. Train programmer
5. **People who accept tasks from computer users, processes it and then return the user there complete tasks are considered as**
   a. Terminal control staff
   b. Data control staff
   c. Peripheral control staff
   d. Librarian control staff
6. **Person who keeps the Disc and Tape files up to date for computer installation is called**
   a. File librarian
   b. Punch librarian
   c. Peripheral librarian
   d. Terminal librarian
7. **Type of programmer who write Programs for a specific user to carry out special instructions is classified as**
   a. Applications programmer
   b. Analysis programmer
   c. Train programmers
   d. Design programmers
8. **A beam is said to be of Uniform strength, if _____**
   a. B.M. is same throughout the beam
   b. Shear stress is the same through the beam
   c. Deflection is the same throughout the beam
   d. Bending stress is the same at every section along its longitudinal axis
9. **Stress in a beam due to simple bending is _____**
   a. Directly proportional
   b. Inversely proportional
   c. Curvilinearly related
   d. None of the mentioned

10. Which stress comes when there is an Eccentric load applied?

a. Shear stress
b. Bending stress
c. Tensile stress
d. Thermal stress

ANSWERS

1 (b)	2 (a)	3 (a)	4 (b)	5 (b)	6 (a)
7 (a)	8 (d)	9 (a)	10 (b)		

2 Continuum Mechanics Problems

OBJECTIVES

After studying this chapter, you will be able to know:

- Continuum Mechanics Problems
- Continuum Mechanics Equations
- Classification of Partial Differential Equations
- Methods for Solving Harmonic and Biharmonic Equations

1. INTRODUCTION

Continuum mechanics is a branch of mechanics that deals with the mechanical behavior of materials modeled as a continuous mass rather than as discrete particles. The French mathematician Augustin-Louis Cauchy was the first to formulate such models in the 19th century. Continuum mechanics studies the response of materials to different loading conditions. Its subject matter can be divided into two main parts: (1) general principles common to all media and (2) constitutive equations defining idealized materials. The general principles are axioms considered to be self-evident from our experience with the physical world, such as conservation of mass; the balance of linear momentum, moment of momentum, and energy; and the entropy inequality law. Mathematically, there are two equivalent forms of the general principles: (1) the integral form, formulated for a finite volume of material in the continuum, and (2) the field equations for differential volume of material (particles) at every point of the field of interest. Field equations are often derived from the integral form. They can also be derived directly from the free body of a differential volume. The latter approach seems to better suit beginners. In this text both approaches are presented. Field equations are important wherever the variations of the variables in the field are either of interest by themselves or are needed to get the desired information. On the other hand, the integral forms of conservation laws lend themselves readily to certain approximate solutions.

The second major part of the theory of continuum mechanics concerns the "constitutive equations" that are used to define idealized materials. Idealized materials represent certain aspects of the mechanical behaviors of natural materials. For example, for many materials, under restricted conditions, the deformation caused by the application of loads disappears with the removal of the loads. This aspect of material behaviors is represented by the constitutive

equation of an elastic body. Under even more restricted conditions, the state of stress at a point depends linearly on the change of lengths and angles suffered by elements at the point measured from the state where the external and internal forces vanish.

The previous expression defines the linearly elastic solid. Another example is supplied by the classical definition of viscosity, which is based on the assumption that the state of stress depends linearly on the instantaneous rates of change of lengths and angles. Such a constitutive equation defines the linearly viscous fluid. The mechanical behaviors of real materials vary not only from material to material but also with different loading conditions for a given material. This leads to the formulation of many constitutive equations defining the many different aspects of material behaviors.

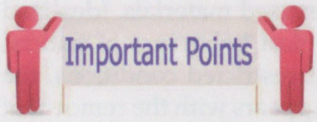
2. CONTINUUM MECHANICS EQUATIONS

Continuum mechanics deals with the behavior of materials that can be approximated as continuous for certain length and time scales. The equations that govern the mechanics of such materials include the balance laws for mass, momentum, and energy. Kinematic relations and constitutive equations are needed to complete the system of governing equations. Physical restrictions on the form of the constitutive relations can be applied by requiring that the second law of thermodynamics be satisfied under all conditions. In the continuum mechanics of solids, the second law of thermodynamics is satisfied if the Clausius–Duhem form of the entropy inequality is satisfied.

The balance laws express the idea that the rate of change of a quantity (mass, momentum, energy) in a volume must arise from three causes:

1. The physical quantity itself flows through the surface that bounds the volume,
2. There is a source of the physical quantity on the surface of the volume, or/and,
3. There is a source of the physical quantity inside the volume.

Let Ω be the body (an open subset of Euclidean space) and let $\partial\Omega$ be its surface (the boundary of Ω).



The equations that govern the mechanics of such materials include the balance laws for mass, momentum, and energy.

Let the motion of material points in the body be described by the map

$$x = x(X) = x\,(X)$$

where X is the position of a point in the initial configuration and x is the location of the same point in the deformed configuration.

The deformation gradient is given by

$$\mathbf{F} = \frac{\partial \mathbf{x}}{\partial \mathbf{X}} = \nabla \mathbf{x}\ .$$

Balance Laws

Let be a physical quantity that is flowing through the body. Let be sources on the surface of the body and let be sources inside the body. Let be the outward unit normal to the surface $\partial\Omega$. Let v (x, t) be the flow velocity of the physical particles that carry the physical quantity that is flowing. Also, let the speed at which the bounding surface $\partial\Omega$ is moving be un (in the direction n).

Then, balance laws can be expressed in the general form

$$\frac{d}{dt}\left[\int_\Omega f(\mathbf{x},t)dV\right] = \int_{\partial\Omega} f(\mathbf{x},t)[u_n(\mathbf{x},t) - \mathbf{v}(\mathbf{x},t)\cdot\mathbf{n}(\mathbf{x},t)]dA + \int_{\partial\Omega} g(\mathbf{x},t)dA + \int_\Omega h(\mathbf{x},t)dV\ .$$

The functions (x,t), g (x,t), and h (x,t) can be scalar valued, vector valued, or tensor valued - depending on the physical quantity that the balance equation deals with. If there are internal boundaries in the body, jump discontinuities also need to be specified in the balance laws.

If we take the Eulerian point of view, it can be shown that the balance laws of mass, momentum, and energy for a solid can be written as (assuming the source term is zero for the mass and angular momentum equations)

$$\dot{\rho} + \rho\nabla\cdot\mathbf{v} = 0 \qquad \text{Balance of Mass}$$

$$\rho\dot{\mathbf{v}} - \nabla\cdot\sigma - \rho\mathbf{b} = 0 \qquad \text{Balance of Linear Momentum (Cauchy's first law of motion)}$$

$$\acute{\mathbf{o}} = \acute{\mathbf{o}}^{T} \qquad \text{Balance of Angular Momentum (Cauchy's second law of motion)}$$

$$\rho\dot{e} - \acute{\mathbf{o}} : (\nabla\mathbf{v}) + \nabla\cdot\mathbf{q} - \rho s = 0 \qquad \text{Balance of Energy.}$$

In the above equations $\rho(x,t)$ is the mass density (current), ρ is the material time derivative of ρ is the particle velocity, v is the material time derivative of $\sigma(x,t)$ is the Cauchy stress tensor, b(x,t) is the body force density, e(x,t) is the

internal energy per unit mass, e is the material time derivative of eq(x,t) is the heat flux vector, and s(x,t) is an energy source per unit mass.

Clausius–Duhem Inequality

The Clausius–Duhem inequality can be used to express the second law of thermodynamics for elastic-plastic materials. This inequality is a statement concerning the irreversibility of natural processes, especially when energy dissipation is involved.

Just like in the balance laws in the previous section, we assume that there is a flux of a quantity, a source of the quantity, and an internal density of the quantity per unit mass. The quantity of interest in this case is the entropy. Thus, we assume that there is an entropy flux, an entropy source, an internal mass density ρ ρand an internal specific entropy (i.e. entropy per unit mass) η in the region of interest.

Let Ω be such a region and let ∂Ω be its boundary. Then the second law of thermodynamics states that the rate of increase of η in this region is greater than or equal to the sum of that supplied to Ω (as a flux or from internal sources) and the change of the internal entropy density ρη due to material flowing in and out of the region.

Let ∂Ω move with a flow velocity and let particles inside Ω have velocities v. Let n be the unit outward normal to the surface ∂Ω. Let ρ be the density of matter in the region, be the entropy flux at the surface, and r be the entropy source per unit mass. Then the entropy inequality may be written as

$$\frac{d}{dt}\left(\int_\Omega \rho\eta\, dV\right) \geq \int_{\partial\Omega} \rho\eta\,(u_n - \mathbf{v}\cdot\mathbf{n})\,dA + \int_{\partial\Omega} \bar{q}\, dA + \int_\Omega \rho r\, dV.$$

The scalar entropy flux can be related to the vector flux at the surface by the relation $\bar{q} = -\boldsymbol{\theta}(\mathbf{x})\cdot\mathbf{n}$. Under the assumption of incrementally isothermal conditions, we have

$$\boldsymbol{\theta}(\mathbf{x}) = \frac{\mathbf{q}(\mathbf{x})}{T}\ ;\ r = \frac{s}{T}$$

where q is the heat flux vector, s is an energy source per unit mass, and T is the absolute temperature of a material point at x at time t.

We then have the Clausius–Duhem inequality in integral form:

$$\frac{d}{dt}\left(\int_\Omega \rho\eta\, dV\right) \geq \int_{\partial\Omega} \rho\eta\,(u_n - \mathbf{v}\cdot\mathbf{n})\,dA - \int_{\partial\Omega} \frac{\mathbf{q}\cdot\mathbf{n}}{T} dA + \int_\Omega \frac{\rho s}{T} dV.$$

We can show that the entropy inequality may be written in differential form as

$$\rho\dot{\eta} \geq -\nabla\cdot\left(\frac{\mathbf{q}}{T}\right)+\frac{\rho s}{T}.$$

In terms of the Cauchy stress and the internal energy, the Clausius–Duhem inequality may be written as

$$\rho\,(\dot{e}-T\dot{\eta})-\boldsymbol{\sigma}:\nabla\mathbf{v} \leq -\frac{\mathbf{q}\cdot\nabla T}{T}.$$

Forces in a Continuum

Continuum mechanics deals with deformable bodies, as opposed to rigid bodies. A solid is a deformable body that possesses shear strength, *sc.* a solid can support shear forces (forces parallel to the material surface on which they act). Fluids, on the other hand, do not sustain shear forces. For the study of the mechanical behavior of solids and fluids these are assumed to be continuous bodies, which means that the matter fills the entire region of space it occupies, despite the fact that matter is made of atoms, has voids, and is discrete. Therefore, when continuum mechanics refers to a point or particle in a continuous body it does not describe a point in the interatomic space or an atomic particle, rather an idealized part of the body occupying that point.

Following the classical dynamics of Newton and Euler, the motion of a material body is produced by the action of externally applied forces which are assumed to be of two kinds: surface forces F_c and body forces F_B. Thus, the total force F applied to a body or to a portion of the body can be expressed as:

$$\mathcal{F} = \mathbf{F}_C + \mathbf{F}_B$$

Surface Forces

Surface forces or contact forces, expressed as force per unit area, can act either on the bounding surface of the body, as a result of mechanical contact with other bodies, or on imaginary internal surfaces that bound portions of the body, as a result of the mechanical interaction between the parts of the body to either side of the surface (Euler-Cauchy's stress principle). When a body is acted upon by external contact forces, internal contact forces are then transmitted from point to point inside the body to balance their action, according to Newton's third law of motion of conservation of linear momentum and angular momentum (for continuous bodies these laws are called the Euler's equations of motion). The internal contact forces are related



For the study of the mechanical behavior of solids and fluids these are assumed to be continuous bodies, which means that the matter fills the entire region of space it occupies, despite the fact that matter is made of atoms, has voids, and is discrete.

to the body's deformation through constitutive equations. The internal contact forces may be mathematically described by how they relate to the motion of the body, independent of the body's material makeup.

The distribution of internal contact forces throughout the volume of the body is assumed to be continuous. Therefore, there exists a *contact force* density or Cauchy traction field T (n,x, t,) that represents this distribution in a particular configuration of the body at a given time t. It is not a vector field because it depends not only on the position x of a particular material point, but also on the local orientation of the surface element as defined by its normal vector n.

Any differential area dS with normal vector n of a given internal surface area S, bounding a portion of the body, experiences a contact force dF_c arising from the contact between both portions of the body on each side of S, and it is given by

$$\mathbf{dF}_C = \mathbf{T}^{(n)}\,dS$$

where $T^{(n)}$ is the surface traction, also called *stress vector*, *traction*, or *traction vector*. The stress vector is a frame-indifferent vector.

The total contact force on the particular internal surface S is then expressed as the sum (surface integral) of the contact forces on all differential surfaces dS:

$$\mathbf{F}_C = \int_S \mathbf{T}^{(n)}\,dS$$

In continuum mechanics a body is considered stress-free if the only forces present are those inter-atomic forces (ionic, metallic, and van der Waals forces) required to hold the body together and to keep its shape in the absence of all external influences, including gravitational attraction. Stresses generated during manufacture of the body to a specific configuration are also excluded when considering stresses in a body. Therefore, the stresses considered in continuum mechanics are only those produced by deformation of the body, *sc.* only relative changes in stress are considered, not the absolute values of stress.

Body Forces

Body forces are forces originating from sources outside of the body that act on the volume (or mass) of the body. Saying that body forces are due to outside sources implies that the interaction between different parts of the body (internal forces) are manifested through the contact forces alone. These forces arise from the presence of the body in force fields, *e.g.* gravitational field (gravitational forces) or electromagnetic field (electromagnetic forces), or from inertial forces when bodies are in motion. As the mass of a continuous body is assumed to be continuously distributed, any force originating from the mass is also continuously distributed. Thus, body forces are specified by vector fields

which are assumed to be continuous over the entire volume of the body, *i.e.* acting on every point in it. Body forces are represented by a body force density b(x, t) (per unit of mass), which is a frame-indifferent vector field.

In the case of gravitational forces, the intensity of the force depends on, or is proportional to, the mass density $\rho(x,t)$ of the material, and it is specified in terms of force per unit mass (b_i) or per unit volume (p_i). These two specifications are related through the material density by the equation $\rho b_i = p_i$. Similarly, the intensity of electromagnetic forces depends upon the strength (electric charge) of the electromagnetic field.

The total body force applied to a continuous body is expressed as

$$\mathbf{F}_B = \int_V \mathbf{b}\,dm = \int_V \rho\mathbf{b}\,dV$$

Body forces and contact forces acting on the body lead to corresponding moments of force (torques) relative to a given point. Thus, the total applied torque M about the origin is given by

$$M = M_c + M_B$$

In certain situations, not commonly considered in the analysis of the mechanical behavior of materials, it becomes necessary to include two other types of forces: these are couple stresses (surface couples, contact torques) and body moments. Couple stresses are moments per unit area applied on a surface. Body moments, or body couples, are moments per unit volume or per unit mass applied to the volume of the body. Both are important in the analysis of stress for a polarized dielectric solid under the action of an electric field, materials where the molecular structure is taken into consideration (e.g. bones), solids under the action of an external magnetic field, and the dislocation theory of metals.

Materials that exhibit body couples and couple stresses in addition to moments produced exclusively by forces are called polar materials. Non-polar materials are then those materials with only moments of forces. In the classical branches of continuum mechanics the development of the theory of stresses is based on non-polar materials.

Thus, the sum of all applied forces and torques (with respect to the origin of the coordinate system) in the body can be given by

$$\mathcal{F} = \int_V \mathbf{a}\,dm = \int_S \mathbf{T}\,dS + \int_V \rho\mathbf{b}\,dV$$

$$\mathcal{M} = \int_S \mathbf{r}\times\mathbf{T}\,dS + \int_V \mathbf{r}\times\rho\mathbf{b}\,dV$$

IMPORTANT

As the mass of a continuous body is assumed to be continuously distributed, any force originating from the mass is also continuously distributed.

3. CLASSIFICATION OF PARTIAL DIFFERENTIAL EQUATIONS

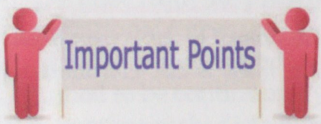
A partial differential equation (or briefly a PDE) is a mathematical equation that involves two or more independent variables, an unknown function (dependent on those variables), and partial derivatives of the unknown function with respect to the independent variables. The order of a partial differential equation is the order of the highest derivative involved. A solution (or a particular solution) to a partial differential equation is a function that solves the equation or, in other words, turns it into an identity when substituted into the equation. A solution is called general if it contains all particular solutions of the equation concerned.

The term exact solution is often used for second- and higher-order nonlinear PDEs to denote a particular solution. Partial differential equations are used to mathematically formulate, and thus aid the solution of, physical and other problems involving functions of several variables, such as the propagation of heat or sound, fluid flow, elasticity, electrostatics, electrodynamics, etc.

Partial differential equations are ubiquitous in mathematically-oriented scientific fields, such as physics and engineering. For instance, they are foundational in the modern scientific understanding of sound, heat, diffusion, electrostatics, electrodynamics, fluid dynamics, elasticity, general relativity, and quantum mechanics. They also arise from many purely mathematical considerations, such as differential geometry and the calculus of variations; among other notable applications, they are the fundamental tool in the proof of the Poincaré conjecture from geometric topology.

Partly due to this variety of sources, there is a wide spectrum of different types of partial differential equations, and methods have been developed for dealing with many of the individual equations which arise. As such, it is usually acknowledged that there is no "general theory" of partial differential equations, with specialist knowledge being somewhat divided between several essentially distinct subfields.

Ordinary differential equations form a subclass of partial differential equations, corresponding to functions of a single variable. Stochastic partial differential equations and nonlocal equations are, as of 2020, particularly widely studied extensions of the "PDE" notion. More classical topics, on which there is still much active research, include elliptic and parabolic partial differential equations, fluid mechanics, Boltzmann equations, and dispersive partial differential equations.



A solution (or a particular solution) to a partial differential equation is a function that solves the equation or, in other words, turns it into an identity when substituted into the equation.

Notation

When writing PDEs, it is common to denote partial derivatives using subscripts. For example:

$$u_x = \frac{\partial u}{\partial x}, \quad u_{xx} = \frac{\partial^2 u}{\partial x^2}, \quad u_{xy} = \frac{\partial^2 u}{\partial y \partial x} = \frac{\partial}{\partial y}\left(\frac{\partial u}{\partial x}\right).$$

In the general situation that u is a function of n variables, then u_i denotes the first partial derivative relative to the i'th input, u_{ij} denotes the second partial derivative relative to the i'th and j'th inputs, and so on.

The Greek letter Δ denotes the Laplace operator; if u is a function of n variables, then

$$\Delta u = u_{11} + u_{22} + \cdots + u_{nn}.$$

In the physics literature, the Laplace operator is often denoted by ∇^2; in the mathematics literature, $\nabla^2 u$ may also denote the hessian matrix of u.

Equations of first order

In mathematics, a first-order partial differential equation is a partial differential equation that involves only first derivatives of the unknown function of n variables. The equation takes the form

$$F(x_1, \ldots, x_n, u, u_{x_1}, \ldots u_{x_n}) = 0.$$

Such equations arise in the construction of characteristic surfaces for hyperbolic partial differential equations, in the calculus of variations, in some geometrical problems, and in simple models for gas dynamics whose solution involves the method of characteristics. If a family of solutions of a single first-order partial differential equation can be found, then additional solutions may be obtained by forming envelopes of solutions in that family. In a related procedure, general solutions may be obtained by integrating families of ordinary differential equations.

Linear and Nonlinear Equations

A PDE is called **linear** if it is linear in the unknown and its derivatives. For example, for a function u of x and y, a second order linear PDE is of the form

$$a_1(x,y)u_{xx} + a_2(x,y)u_{xy} + a_3(x,y)u_{yx} + a_4(x,y)u_{yy}$$
$$+a_5(x,y)u_x + a_6(x,y)u_y + a_7(x,y)u = f(x,y)$$

where a_i and f are functions of the independent variables only. (Often the mixed-partial derivatives u_{xy} and u_{yx} will be equated, but this is not required for the discussion of linearity.) If the a_i are constants (independent

of x and y) then the PDE is called linear with constant coefficients. If f is zero everywhere then the linear PDE is homogeneous, otherwise it is inhomogeneous. (This is separate from asymptotic homogenization, which studies the effects of high-frequency oscillations in the coefficients upon solutions to PDEs.)

Nearest to linear PDEs are semilinear PDEs, where the highest order derivatives appear only as linear terms, with coefficients that are functions of the independent variables only. The lower order derivatives and the unknown function may appear arbitrarily otherwise. For example, a general second order semilinear PDE in two variables is

$$a_1(x,y)u_{xx} + a_2(x,y)u_{xy} + a_3(x,y)u_{yx}$$
$$+a_4(x,y)u_{yy} + f(u_x, u_y, u, x, y) = 0$$

In a quasilinear PDE the highest order derivatives likewise appear only as linear terms, but with coefficients possibly functions of the unknown and lower-order derivatives:

$$a_1(u_x, u_y, u, x, y)u_{xx} + a_2(u_x, u_y, u, x, y)u_{xy} +$$
$$a_3(u_x, u_y, u, x, y)u_{yx} + a_4(u_x, u_y, u, x, y)u_{yy} + f(u_x, u_y, u, x, y) = 0$$

Many of the fundamental PDEs in physics are quasilinear, such as the Einstein equations of general relativity and the Navier–Stokes equations describing fluid motion. A PDE without any linearity properties is called fully nonlinear, and possesses nonlinearities on one or more of the highest-order derivatives. An example is the Monge–Ampère equation, which arises in differential geometry.

Linear Equations of Second Order

Elliptic, parabolic, and hyperbolic partial differential equations of order two have been widely studied since the beginning of the twentieth century. However, there are many other important types of PDE, including the Korteweg–de Vries equation. There are also hybrids such as the Euler–Tricomi equation, which vary from elliptic to hyperbolic for different regions of the domain. There are also important extensions of these basic types to higher-order PDE, but such knowledge is more specialized.

The elliptic/parabolic/hyperbolic classification provides a guide to appropriate initial and boundary conditions and to the smoothness of the solutions. Assuming uxy = uyx, the general linear second-order PDE in two independent variables has the form

$$Au_{xx} + 2Bu_{xy} + Cu_{yy} + \cdots (\text{lower order terms}) = 0,$$

where the coefficients A, B, C... may depend upon x and y. If $A^2 + B^2 + C^2 > 0$ over a region of the xy-plane, the PDE is second-order in that region. This form is analogous to the equation for a conic section:

$$Ax^2 + 2Bxy + Cy^2 + \cdots = 0.$$

More precisely, replacing ∂_x by X, and likewise for other variables (formally this is done by a Fourier transform), converts a constant-coefficient PDE into a polynomial of the same degree, with the terms of the highest degree (a homogeneous polynomial, here a quadratic form) being most significant for the classification.

Just as one classifies conic sections and quadratic forms into parabolic, hyperbolic, and elliptic based on the discriminant $B^2 - 4AC$, the same can be done for a second-order PDE at a given point. However, the discriminant in a PDE is given by $B^2 - AC$ due to the convention of the xy term being 2B rather than B; formally, the discriminant (of the associated quadratic form) is $(2B)2 - 4AC = 4(B^2 - AC)$, with the factor of 4 dropped for simplicity.

1. $B^2 - AC < 0$ (elliptic partial differential equation): Solutions of elliptic PDEs are as smooth as the coefficients allow, within the interior of the region where the equation and solutions are defined. For example, solutions of Laplace's equation are analytic within the domain where they are defined, but solutions may assume boundary values that are not smooth. The motion of a fluid at subsonic speeds can be approximated with elliptic PDEs, and the Euler–Tricomi equation is elliptic where $x < 0$.
2. $B^2 - AC = 0$ (parabolic partial differential equation): Equations that are parabolic at every point can be transformed into a form analogous to the heat equation by a change of independent variables. Solutions smooth out as the transformed time variable increases. The Euler–Tricomi equation has parabolic type on the line where $x = 0$.
3. $B^2 - AC > 0$ (hyperbolic partial differential equation): hyperbolic equations retain any discontinuities of functions or derivatives in the initial data. An example is the wave equation. The motion of a fluid at supersonic speeds can be approximated with hyperbolic PDEs, and the Euler–Tricomi equation is hyperbolic where $x > 0$.

If there are n independent variables x_1, x_2, ..., x_n, a general linear partial differential equation of second order has the form

$$Lu = \sum_{i=1}^{n} \sum_{j=1}^{n} a_{i,j} \frac{\partial^2 u}{\partial x_i \partial x_j} \quad \text{plus lower-order terms} = 0.$$

The classification depends upon the signature of the eigenvalues of the coefficient matrix $a_{i,j}$.

1. **Elliptic:** the eigenvalues are all positive or all negative.
2. **Parabolic:** the eigenvalues are all positive or all negative, save one that is zero.
3. **Hyperbolic:** there is only one negative eigenvalue and all the rest are positive, or there is only one positive eigenvalue and all the rest are negative.
4. **Ultrahyperbolic:** there is more than one positive eigenvalue and more than one negative eigenvalue, and there are no zero eigenvalues. There is only a limited theory for ultrahyperbolic equations.

Systems of First-Order Equations and Characteristic Surfaces

The classification of partial differential equations can be extended to systems of first-order equations, where the unknown u is now a vector with m components, and the coefficient matrices A_ν are m by m matrices for $\nu = 1, 2, \ldots, n$. The partial differential equation takes the form

$$Lu = \sum_{\nu=1}^{n} A_\nu \frac{\partial u}{\partial x_\nu} + B = 0,$$

where the coefficient matrices A_ν and the vector B may depend upon x and u. If a hypersurface S is given in the implicit form

$$\varphi(x_1, x_2, \ldots, x_n) = 0,$$

where φ has a non-zero gradient, then S is a characteristic surface for the operator L at a given point if the characteristic form vanishes:

$$Q\left(\frac{\partial \varphi}{\partial x_1}, \ldots \frac{\partial \varphi}{\partial x_n}\right) = \det\left[\sum_{\nu=1}^{n} A_\nu \frac{\partial \varphi}{\partial x_\nu}\right] = 0.$$

The geometric interpretation of this condition is as follows: if data for u are prescribed on the surface S, then it may be possible to determine the normal derivative of u on S from the differential equation. If the data on S and the differential equation determine the normal derivative of u on S, then S is non-characteristic. If the data on S and the differential equation *do not* determine the normal derivative of u on S, then the surface is characteristic, and the differential equation restricts the data on S: the differential equation is *internal* to S.

1. A first-order system Lu = 0 is elliptic if no surface is characteristic for L: the values of u on S and the differential equation always determine the normal derivative of u on S.
2. A first-order system is hyperbolic at a point if there is a spacelike surface S with normal ξ at that point. This means that, given any

non-trivial vector η orthogonal to ξ, and a scalar multiplier λ, the equation $Q(\lambda\xi + \eta) = 0$ has m real roots $\lambda_1, \lambda_2, \ldots, \lambda_m$. The system is strictly hyperbolic if these roots are always distinct. The geometrical interpretation of this condition is as follows: the characteristic form $Q(\zeta) = 0$ defines a cone (the normal cone) with homogeneous coordinates ζ. In the hyperbolic case, this cone has m sheets, and the axis $\zeta = \lambda\xi$ runs inside these sheets: it does not intersect any of them. But when displaced from the origin by η, this axis intersects every sheet. In the elliptic case, the normal cone has no real sheets.

Numerical Solutions

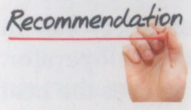
The three most widely used numerical methods to solve PDEs are the finite element method (FEM), finite volume methods (FVM) and finite difference methods (FDM), as well other kind of methods called Meshfree methods, which were made to solve problems where the aforementioned methods are limited. The FEM has a prominent position among these methods and especially its exceptionally efficient higher-order version hp-FEM. Other hybrid versions of FEM and Meshfree methods include the generalized finite element method (GFEM), extended finite element method (XFEM), spectral finite element method (SFEM), meshfree finite element method, discontinuous Galerkin finite element method (DGFEM), Element-Free Galerkin Method (EFGM), Interpolating Element-Free Galerkin Method (IEFGM), etc.

Finite Element Method

The finite element method (FEM) (its practical application often known as finite element analysis (FEA)) is a numerical technique for finding approximate solutions of partial differential equations (PDE) as well as of integral equations. The solution approach is based either on eliminating the differential equation completely (steady state problems), or rendering the PDE into an approximating system of ordinary differential equations, which are then numerically integrated using standard techniques such as Euler's method, Runge–Kutta, etc.

Finite Difference Method

Finite-difference methods are numerical methods for approximating the solutions to differential equations using finite difference equations to approximate derivatives.



The solution approach is based either on eliminating the differential equation completely (steady state problems), or rendering the PDE into an approximating system of ordinary differential equations.

Finite Volume Method

Similar to the finite difference method or finite element method, values are calculated at discrete places on a meshed geometry. "Finite volume" refers to the small volume surrounding each node point on a mesh. In the finite volume method, surface integrals in a partial differential equation that contain a divergence term are converted to volume integrals, using the divergence theorem. These terms are then evaluated as fluxes at the surfaces of each finite volume. Because the flux entering a given volume is identical to that leaving the adjacent volume, these methods conserve mass by design.

Analytical Solutions

Separation of variables

Linear PDEs can be reduced to systems of ordinary differential equations by the important technique of separation of variables. This technique rests on a characteristic of solutions to differential equations: if one can find any solution that solves the equation and satisfies the boundary conditions, then it is the solution (this also applies to ODEs). We assume as an ansatz that the dependence of a solution on the parameters space and time can be written as a product of terms that each depend on a single parameter, and then see if this can be made to solve the problem.

In the method of separation of variables, one reduces a PDE to a PDE in fewer variables, which is an ordinary differential equation if in one variable – these are in turn easier to solve.

This is possible for simple PDEs, which are called separable partial differential equations, and the domain is generally a rectangle (a product of intervals). Separable PDEs correspond to diagonal matrices – thinking of "the value for fixed x" as a coordinate, each coordinate can be understood separately. This generalizes to the method of characteristics, and is also used in integral transforms.

Method of Characteristics

In special cases, one can find characteristic curves on which the equation reduces to an ODE – changing coordinates in the domain to straighten these curves allows separation of variables, and is called the method of characteristics. More generally, one may find characteristic surfaces.

Integral Transform

An integral transform may transform the PDE to a simpler one, in particular, a separable PDE. This corresponds to diagonalizing an operator. An important example of this is Fourier analysis, which diagonalizes the heat equation using the eigenbasis of sinusoidal waves.

If the domain is finite or periodic, an infinite sum of solutions such as a Fourier series is appropriate, but an integral of solutions such as a Fourier integral is generally required for infinite domains. The solution for a point source for the heat equation given above is an example of the use of a Fourier integral.

Change of Variables

Often a PDE can be reduced to a simpler form with a known solution by a suitable change of variables. For example, the Black–Scholes PDE

$$\frac{\partial V}{\partial t} + \tfrac{1}{2}\sigma^2 S^2 \frac{\partial^2 V}{\partial S^2} + rS\frac{\partial V}{\partial S} - rV = 0$$

is reducible to the heat equation

$$\frac{\partial u}{\partial \tau} = \frac{\partial^2 u}{\partial x^2}$$

by the change of variables

$$V(S,t) = Kv(x,\tau),$$
$$x = \ln\left(\tfrac{S}{K}\right),$$
$$\tau = \tfrac{1}{2}\sigma^2(T-t),$$
$$v(x,\tau) = e^{-\alpha x - \beta\tau}u(x,\tau).$$

Fundamental Solution

Inhomogeneous equations can often be solved (for constant coefficient PDEs, always be solved) by finding the fundamental solution (the solution for a point source), then taking the convolution with the boundary conditions to get the solution.

This is analogous in signal processing to understanding a filter by its impulse response.

Superposition Principle

The superposition principle applies to any linear system, including linear systems of PDEs. A common visualization of this concept is the interaction of two waves in phase being combined to result in a greater amplitude, for example $\sin x + \sin x = 2 \sin x$. The same principle can be observed in PDEs where the solutions may be real or complex and additive. superposition If u_1 and u_2 are solutions of linear PDE in some function space R, then $u = c_1u_1 + c_2u_2$ with any constants c_1 and c_2 are also a solution of that PDE in the same function space.

Methods for Non-Linear Equations

There are no generally applicable methods to solve nonlinear PDEs. Still, existence and uniqueness results (such as the Cauchy–Kowalevski theorem) are often possible, as are proofs of important qualitative and quantitative properties of solutions (getting these results is a major part of analysis). Computational solution to the nonlinear PDEs, the split-step method, exist for specific equations like nonlinear Schrödinger equation.

Nevertheless, some techniques can be used for several types of equations. The h-principle is the most powerful method to solve underdetermined equations. The Riquier–Janet theory is an effective method for obtaining information about many analytic overdetermined systems.

The method of characteristics can be used in some very special cases to solve partial differential equations. In some cases, a PDE can be solved via perturbation analysis in which the solution is considered to be a correction to an equation with a known solution. Alternatives are numerical analysis techniques from simple finite difference schemes to the more mature multigrid and finite element methods. Many interesting problems in science and engineering are solved in this way using computers, sometimes high performance supercomputers.

Lie Group Method

From 1870 Sophus Lie's work put the theory of differential equations on a more satisfactory foundation. He showed that the integration theories of the older mathematicians can, by the introduction of what are now called Lie groups, be referred, to a common source; and that ordinary differential equations which admit the same infinitesimal transformations present comparable difficulties of integration. He also emphasized the subject of transformations of contact.

A general approach to solving PDEs uses the symmetry property of differential equations, the continuous infinitesimal transformations of solutions to solutions (Lie theory). Continuous group theory, Lie algebras and differential geometry are used to understand the structure of linear and nonlinear partial differential equations for generating integrable equations, to find its Lax pairs, recursion operators, Bäcklund transform and finally finding exact analytic solutions to the PDE.

Symmetry methods have been recognized to study differential equations arising in mathematics, physics, engineering, and many other disciplines.

Semianalytical Methods

The Adomian decomposition method, the Lyapunov artificial small parameter method, and his homotopy perturbation method are all special cases of the more general homotopy analysis method. These are series expansion methods, and except for the Lyapunov method, are independent of small physical parameters as compared to the well known perturbation theory, thus giving these methods greater flexibility and solution generality.

Elliptic, Hyperbolic, and Parabolic PDEs

We usually come across three-types of second-order PDEs in mechanics. These are classified as elliptic, hyperbolic, and parabolic.

The equations of elasticity (without inertial terms) are elliptic PDEs. Hyperbolic PDEs describe wave propagation phenomena. The heat conduction equation is an example of a parabolic PDE.

Each type of PDE has certain characteristics that help determine if a particular finite element approach is appropriate to the problem being described by the PDE. Interestingly, just knowing the type of PDE can give us insight into how smooth the solution is, how fast information propagates, and the effect of initial and boundary conditions.

In hyperbolic PDEs, the smoothness of the solution depends on the smoothness of the initial and boundary conditions. For instance, if there is a jump in the data at the start or at the boundaries, then the jump will propagate as a shock in the solution. If, in addition, the PDE is nonlinear, then shocks may develop even though the initial conditions and the boundary conditions are smooth. In a system modeled with a hyperbolic PDE, information travels at a finite speed referred to as the wavespeed. Information is not transmitted until the wave arrives.

In contrast, the solutions of elliptic PDEs are always smooth, even if the initial and boundary conditions are rough (though there may be singularities at sharp corners). In addition, boundary data at any point affect the solution at all points in the domain.

Parabolic PDEs are usually time dependent and represent diffusion-like processes. Solutions are smooth in space but may possess singularities. However, information travels at infinite speed in a parabolic system.



Interestingly, just knowing the type of PDE can give us insight into how smooth the solution is, how fast information propagates, and the effect of initial and boundary conditions.

Suppose we have a second-order PDE of the form

$$a(x_1,x_2)\frac{\partial^2 u}{\partial x_1^2}+b(x_1,x_2)\frac{\partial^2 u}{\partial x_1 \partial x_2}+c(x_1,x_2)\frac{\partial^2 u}{\partial x_2^2}$$

$$+d(x_1,x_2)\frac{\partial u}{\partial x_1}+e(x_1,x_2)\frac{\partial u}{\partial x_2}+f(x_1,x_2)u=g(x_1,x_2)$$

Then, the PDE is called elliptic if

$$b^2 - 4ac < 0$$

An example is

$$\frac{\partial^2 u}{\partial x_1^2}+\frac{\partial^2 u}{\partial x_1 \partial x_2}+\frac{\partial^2 u}{\partial x_2^2}=x_1\frac{\partial u}{\partial x_1}$$

The PDE is called hyperbolic if

$$b^2 - 4ac > 0$$

An example is

$$\frac{\partial^2 u}{\partial x_1^2}+3\frac{\partial^2 u}{\partial x_1 \partial x_2}+\frac{\partial^2 u}{\partial x_2^2}=x_1\frac{\partial u}{\partial x_1}$$

The PDE is called parabolic if

$$b^2 - 4ac = 0$$

An example is

$$\frac{\partial^2 u}{\partial x_1^2}+2\frac{\partial^2 u}{\partial x_1 \partial x_2}+\frac{\partial^2 u}{\partial x_2^2}=x_1\frac{\partial u}{\partial x_1}$$

4. METHODS FOR SOLVING HARMONIC AND BIHARMONIC EQUATIONS

In mathematics, the biharmonic equation is a fourth-order partial differential equation which arises in areas of continuum mechanics, including linear elasticity theory and the solution of Stokes flows. Specifically, it is used in the modeling of thin structures that react elastically to external forces.

Notation

It is written as $\nabla^4 \varphi = 0$

or $\nabla^2 \nabla^2 \varphi = 0$

or $\Delta^2 \varphi = 0$

where ∇^4, which is the fourth power of the del operator and the square of the Laplacian operator ∇^2 (or Δ), is known as the biharmonic operator

or the bilaplacian operator. In Cartesian coordinates, it can be written in n {\displaystyle n} dimensions as:

$$\nabla^4\varphi = \sum_{i=1}^{n}\sum_{j=1}^{n}\partial_i\partial_i\partial_j\partial_j\varphi = \left(\sum_{i=1}^{n}\partial_i\partial_i\right)\left(\sum_{j=1}^{n}\partial_j\partial_j\right)\varphi.$$

Because the formula here contains a summation of indices, many mathematicians prefer the notation Δ^2 over ∇^4 because the former makes clear which of the indices of the four nabla operators are contracted over.

For example, in three dimensional Cartesian coordinates the biharmonic equation has the form

$$\frac{\partial^4\varphi}{\partial x^4}+\frac{\partial^4\varphi}{\partial y^4}+\frac{\partial^4\varphi}{\partial z^4}+2\frac{\partial^4\varphi}{\partial x^2\partial y^2}+2\frac{\partial^4\varphi}{\partial y^2\partial z^2}+2\frac{\partial^4\varphi}{\partial x^2\partial z^2}=0.$$

As another example, in n-dimensional Real coordinate space without the origin ($\mathbf{R}^n\setminus 0$),

$$\nabla^4\left(\frac{1}{r}\right)=\frac{3(15-8n+n^2)}{r^5}$$

where

$$r=\sqrt{x_1^2+x_2^2+\cdots+x_n^2}.$$

which shows, for $n=3$ and $n=5$ only, $\frac{1}{r}$ is a solution to the biharmonic equation.

A solution to the biharmonic equation is called a biharmonic function. Any harmonic function is biharmonic, but the converse is not always true.

In two-dimensional polar coordinates, the biharmonic equation is

$$\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial}{\partial r}\left(\frac{1}{r}\frac{\partial}{\partial r}\left(r\frac{\partial\varphi}{\partial r}\right)\right)\right)+\frac{2}{r^2}\frac{\partial^4\varphi}{\partial\theta^2\partial r^2}+\frac{1}{r^4}\frac{\partial^4\varphi}{\partial\theta^4}-\frac{2}{r^3}\frac{\partial^3\varphi}{\partial\theta^2\partial r}+\frac{4}{r^4}\frac{\partial^2\varphi}{\partial\theta^2}=0$$

which can be solved by separation of variables. The result is the Michell solution.

Two-Dimensional Space

The general solution to the 2-dimensional case is

$$xv(x,y) - yu(x,y) + w(x,y)$$

where u(x,y), v(x,y) and w(x,y) are harmonic functions andv(x,y) is a harmonic conjugate of u(x,y).

Just as harmonic functions in 2 variables are closely related to complex analytic functions, so are biharmonic functions in 2 variables. The general form of a biharmonic function in 2 variables can also be written as

$$\mathrm{Im}(\overline{z}f(z)+g(z))$$

where f(z) and g(z) are analytic functions.

Harmonic Equations

The descriptor "harmonic" in the name harmonic function originates from a point on a taut string which is undergoing harmonic motion. The solution to the differential equation for this type of motion can be written in terms of sines and cosines, functions which are thus referred to as harmonics. Fourier analysis involves expanding functions on the unit circle in terms of a series of these harmonics. Considering higher dimensional analogues of the harmonics on the unit n-sphere, one arrives at the spherical harmonics. These functions satisfy Laplace's equation and over time "harmonic" was used to refer to all functions satisfying Laplace's equation.

In mathematics, mathematical physics and the theory of stochastic processes, a harmonic function is a twice continuously differentiable function $f : U \to \mathrm{R}$, where U is an open subset of R^n, that satisfies Laplace's equation, that is,

$$\frac{\partial^2 f}{\partial x_1^2}+\frac{\partial^2 f}{\partial x_2^2}+\cdots+\frac{\partial^2 f}{\partial x_n^2}=0$$

everywhere on U. This is usually written as

$$\nabla^2 f = 0$$

or

$$\nabla f = 0$$

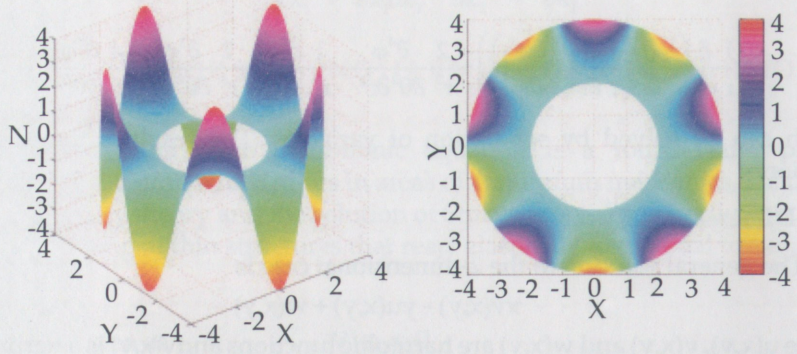
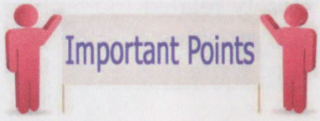


Fig. 2.1 A harmonic function defined on an annulus



A solution to the biharmonic equation is called a biharmonic function. Any harmonic function is biharmonic, but the converse is not always true.

Examples of Harmonic Functions

Examples of harmonic functions of two variables are:

- The real and imaginary parts of any holomorphic function
- The function $f(x,y) = e^x \sin y$; this is a special case of the example above, as $f(x,y) = \text{Im}\,(e^{x+iy})$, and e^{x+iy} is a holomorphic function.
- The function $f(x,y) = \ln(x^2+y^2)$ defined on $\mathbf{R}^2 \setminus \{0\}$. This can describe the electric potential due to a line charge or the gravity potential due to a long cylindrical mass.

Examples of harmonic functions of three variables are given in the table below with $r^2 = x^2 + y^2 + z^2$: :

Table 2.1 Examples of harmonic functions of three variables

Function	Singularity
$\frac{1}{r}$	Unit point charge at origin
$\frac{X}{r^3}$	x-directed dipole at origin
$-\ln(r^2 - Z^2)$	Line of unit charge density on entire z-axis
$-\ln(r + Z)$	Line of unit charge density on negative z-axis
$\frac{X}{r^2 - Z^2}$	Line of x-directed dipoles on entire z axis
$\frac{X}{r + Z}$	Line of x-directed dipoles on negative z axis

Harmonic functions that arise in physics are determined by their singularities and boundary conditions (such as Dirichlet boundary conditions or Neumann boundary conditions). On regions without boundaries, adding the real or imaginary part of any entire function will produce a harmonic function with the same singularity, so in this case the harmonic function is not determined by its singularities; however, we can make the solution unique in physical situations by requiring that the solution approaches 0 as r approaches infinity. In this case, uniqueness follows by Liouville's theorem.

The singular points of the harmonic functions above are expressed as "charges" and "charge densities" using the terminology of electrostatics, and so the corresponding harmonic function will be proportional to the electrostatic potential due to these charge distributions. Each function above

will yield another harmonic function when multiplied by a constant, rotated, and/or has a constant added. The inversion of each function will yield another harmonic function which has singularities which are the images of the original singularities in a spherical "mirror". Also, the sum of any two harmonic functions will yield another harmonic function.

Finally, examples of harmonic functions of n variables are:

The constant, linear and affine functions on all of R^n (for example, the electric potential between the plates of a capacitor, and the gravity potential of a slab)

The function $f(x_1,...,x_n) = (x_1^2+...x_n^2)^{1-n/2}$ on $R^n \setminus \{0\}$ for $n > 2$.

Connections with complex function theory

The real and imaginary part of any holomorphic function yield harmonic functions on R^2 (these are said to be a pair of harmonic conjugate functions). Conversely, any harmonic function u on an open subset Ω of $\mathbf{R}^2$ is *locally* the real part of a holomorphic function. This is immediately seen observing that, writing $z = x + iy$, the complex function $g(z) := u_x - i\, u_y$ is holomorphic in Ω because it satisfies the Cauchy–Riemann equations. Therefore, g locally has a primitive f, and u is the real part of f up to a constant, as u_x is the real part of $f = g$.

Although, the above correspondence with holomorphic functions only holds for functions of two real variables, harmonic functions in n variables still enjoy a number of properties typical of holomorphic functions. They are (real) analytic; they have a maximum principle and a mean-value principle; a theorem of removal of singularities as well as a Liouville theorem holds for them in analogy to the corresponding theorems in complex functions theory.

Remarks

The set of harmonic functions on a given open set U can be seen as the kernel of the Laplace operator Δ and is therefore a vector space over R: linear combinations of harmonic functions are again harmonic. If f is a harmonic function on U, then all partial derivatives of f are also harmonic functions on U. The Laplace operator Δ and the partial derivative operator will commute on this class of functions. In several ways, the harmonic functions are real analogues to holomorphic functions. All harmonic functions are analytic, that is, they can be locally expressed as power series. This is a general fact about elliptic operators, of which the Laplacian is a major example.

The uniform limit of a convergent sequence of harmonic functions is still harmonic. This is true because every continuous function satisfying the mean value property is harmonic. Consider the sequence on $(-\infty, 0) \times R$ defined by

$$f_n(x,y) = \frac{1}{n}\exp(nx)\cos(ny)$$

This sequence is harmonic and converges uniformly to the zero function; however note that the partial derivatives are not uniformly convergent to the zero function (the derivative of the zero function). This example shows the importance of relying on the mean value property and continuity to argue that the limit is harmonic.

CONCLUSION

A class of biharmonic equations with sign-changing potentials and an indefinite nonlinearity is studied in this chapter. Under some suitable conditions, the existence of nontrivial solutions and the high energy solutions are obtained by using variational methods. Moreover, the phenomenon of concentration of solutions is explored. The results extend the main conclusions in recent literature. Partial differential equations problems are often intrinsically connected to the unconstrained minimization of a quadratic energy functional. The associated Rayleigh–Ritz variational principles provide an attractive setting for the development of finite element methods.

It surveys the most important PDEs with the Dirichlet or Neumann boundary conditions: the Laplace, Poisson, wave, and diffusion equations. The method of separation of variables is discussed for Cartesian, cylindrical, and spherical coordinate systems. Two situations are identified: One with the solution a unique linear combination of basic solutions (normal modes), and one with many individual solutions, each associated with an eigenvalue. Examples of both problem types are given. The angular solutions to central-force PDEs are examined in detail. When in a specific form they are identified as spherical harmonics.

Partial differential equations (PDEs) of hyperbolic/nearly hyperbolic type are of fundamental importance in many areas of applied mathematics and engineering, particularly for applications arising in fluid dynamics and electromagnetics. Typically, solutions to these types of equations exhibit localized phenomena, such as propagating discontinuities and sharp transition layers, and their reliable numerical approximation represents a challenging computational task. Indeed, in order to resolve such localized features in an accurate and efficient manner it is essential to use adaptively refined computational meshes whose construction is governed by sharp a posteriori error bounds.

REFERENCES

- Adomian, G. (1994). Solving Frontier problems of Physics: The decomposition method. Kluwer Academic Publishers. ISBN 9789401582896.
- Batra, R. C. (2006). Elements of Continuum Mechanics. Reston, VA: AIAA.
- Bertram, Albrecht (2012). Elasticity and Plasticity of Large Deformations - An Introduction (Third ed.). Springer. doi:10.1007/978-3-642-24615-9. ISBN 978-3-642-24615-9.

- Chandramouli, P.N (2014). Continuum Mechanics. Yes Dee Publishing Pvt Ltd. ISBN 9789380381398.
- Chen, Youping; James D. Lee; Azim Eskandarian (2009). Meshless Methods in Solid Mechanics (First ed.). Springer New York. ISBN 978-1-4419-2148-2.
- Courant, R. & Hilbert, D. (1962). Methods of Mathematical Physics: Partial Differential Equations. II. New York: Wiley-Interscience.
- Dienes, J. K.; Solem, J. C. (1999). "Nonlinear behavior of some hydrostatically stressed isotropic elastomeric foams". Acta Mechanica. 138 (3–4): 155–162. doi:10.1007/BF01291841. S2CID 120320672.
- Dill, Ellis Harold (2006). Continuum Mechanics: Elasticity, Plasticity, Viscoelasticity. Germany: CRC Press. ISBN 978-0-8493-9779-0.
- Dimitrienko, Yuriy (2011). Nonlinear Continuum Mechanics and Large Inelastic Deformations. Germany: Springer. ISBN 978-94-007-0033-8.
- Drábek, Pavel; Holubová, Gabriela (2007). Elements of partial differential equations ([Online-Ausg.]. ed.). Berlin: de Gruyter. ISBN 9783110191240.
- Eric W Weisstein, CRC Concise Encyclopedia of Mathematics, CRC Press, 2002. ISBN 1-58488-347-2.
- Eringen, A. Cemal (1980). Mechanics of Continua (2nd ed.). Krieger Pub Co. ISBN 978-0-88275-663-9.
- Evans, L. C. (1998), Partial Differential Equations, Providence: American Mathematical Society, ISBN 0-8218-0772-2.
- Fung, Y. C. (1977). A First Course in Continuum Mechanics (2nd ed.). Prentice-Hall, Inc. ISBN 978-0-13-318311-5.
- Garabedian, P. R. (1964). Partial Differential Equations. New York: Wiley. OCLC 527754.
- Gershenfeld, N. (1999), The Nature of Mathematical Modeling (1st ed.), New York: Cambridge University Press, New York, NY, USA, ISBN 0-521-57095-6.
- Gurtin, M. E. (1981). An Introduction to Continuum Mechanics. New York: Academic Press.
- Gustafsson, Bertil (2008). High Order Difference Methods for Time Dependent PDE. Springer Series in Computational Mathematics. 38. Springer. doi:10.1007/978-3-540-74993-6. ISBN 978-3-540-74992-9.
- Hutter, Kolumban; Klaus Jöhnk (2004). Continuum Methods of Physical Modeling. Germany: Springer. ISBN 978-3-540-20619-4.
- Ibragimov, Nail H. (1993), CRC Handbook of Lie Group Analysis of Differential Equations Vol. 1-3, Providence: CRC-Press, ISBN 0-8493-4488-3.
- John, F. (1982), Partial Differential Equations (4th ed.), New York: Springer-Verlag, ISBN 0-387-90609-6.

- Jost, J. (2002), Partial Differential Equations, New York: Springer-Verlag, ISBN 0-387-95428-7.
- Krasil'shchik, I.S.; Lychagin, V.V. & Vinogradov, A.M. (1986), Geometry of Jet Spaces and Nonlinear Partial Differential Equations, Gordon and Breach Science Publishers, New York, London, Paris, Montreux, Tokyo, ISBN 2-88124-051-8.
- Lai, W. Michael; David Rubin; Erhard Krempl (1996). Introduction to Continuum Mechanics (3rd ed.). Elsevier, Inc. ISBN 978-0-7506-2894-5. Archived from the original on 6 February 2009.
- Liao, S.J. (2003), Beyond Perturbation: Introduction to the Homotopy Analysis Method, Boca Raton: Chapman & Hall/ CRC Press, ISBN 1-58488-407-X.
- Lubliner, Jacob (2008). Plasticity Theory (PDF) (Revised ed.). Dover Publications. ISBN 978-0-486-46290-5. Archived from the original (PDF) on 31 March 2010.
- Malvern, Lawrence E. (1969). Introduction to the mechanics of a continuous medium. New Jersey: Prentice-Hall, Inc.
- Mase, G. Thomas; George E. Mase (1999). Continuum Mechanics for Engineers (Second ed.). CRC Press. ISBN 978-0-8493-1855-9.
- Mase, George E. (1970). Continuum Mechanics. McGraw-Hill Professional. ISBN 978-0-07-040663-6.
- Maugin, G. A. (1999). The Thermomechanics of Nonlinear Irreversible Behaviors: An Introduction. Singapore: World Scientific.
- Nemat-Nasser, Sia (2006). Plasticity: A Treatise on Finite Deformation of Heterogeneous Inelastic Materials. Cambridge: Cambridge University Press. ISBN 978-0-521-83979-2.

MCQs

1. **By using the method of separation of variables, the determination of solution to P.D.E. reduces to determination of solution to O.D.E.**
   a. True
   b. False
2. **Separation of variables, in mathematics, is also known as Fourier method.**
   a. False
   b. True
3. **Which of the following equations cannot be solved by using the method of separation of variables?**
   a. Laplace Equation
   b. Helmholtz Equation
   c. Alpha Equation
   d. Biharmonic Equation

4. **The matrix form of the separation of variables is the Kronecker sum.**
   a. True
   b. False
5. **For a partial differential equation, in a function φ (x, y) and two variables x, y, what is the form obtained after separation of variables is applied?**
   a. $\Phi(x, y) = X(x)+Y(y)$
   b. $\Phi(x, y) = X(x)-Y(y)$
   c. $\Phi(x, y) = X(x)Y(y)$
   d. $\Phi(x, y) = X(x)/Y(y)$
6. **The given equation satisfies the Laplace equation.$V = x^2 + y^2 - z^2$. State True/False.**
   a. True
   b. False
7. **In free space, the Poisson equation becomes**
   a. Maxwell equation
   b. Ampere equation
   c. Laplace equation
   d. Steady state equation
8. **If Laplace equation satisfies, then which of the following statements will be true?**
   a. Potential will be zero
   b. Current will be infinite
   c. Resistance will be infinite
   d. Voltage will be same
9. **Suppose the Potential function is a step function. The equation that gets satisfied is**
   a. Laplace equation
   b. Poisson equation
   c. Maxwell equation
   d. Ampere equation

10. **While solving a partial differential equation using a variable separable method, we assume that the function can be written as the product of two functions which depend on one variable only.**
    a. True
    b. False
11. **While solving a partial differential equation using a variable separable method, we equate the ratio to a constant which?**
    a. Can be positive or negative integer or zero
    b. Can be positive or negative rational number or zero
    c. Must be a positive integer
    d. Must be a negative integer
12. **Calculate the charge density when a potential function $x^2 + y^2 + z^2$ is in air(in 10-9 order)**
    a. $1/6\pi$
    b. $6/2\pi$
    c. $12/6\pi$
    d. $10/8\pi$

ANSWERS

1 (a)	2 (b)	3 (c)	4 (a)	5 (c)	6 (a)
7 (c)	8 (b)	9 (a)	10 (a)	11 (b)	12 (a)

3 Finite Element Analysis of Harmonic Problems

OBJECTIVES

After studying this chapter, you will be able to know:

- Derivation of the Element Stiffness Matrix
- Assembly of the Overall Stiffness Matrix
- Comparison With the Finite Difference Approach
- Variational Formulation
- Boundary Conditions
- Solution of the Linear Equations

1. INTRODUCTION

The Finite Element Analysis (FEA) is the simulation of any given physical phenomenon using the numerical technique called Finite Element Method (FEM). Engineers use FEA software to reduce the number of physical prototypes and experiments and optimize components in their design phase to develop better products, faster while saving on expenses. It is necessary to use mathematics to comprehensively understand and quantify any physical phenomena such as structural or fluid behavior, thermal transport, wave propagation, the growth of biological cells, etc. Most of these processes are described using Partial Differential Equations (PDEs). However, for a computer to solve these PDEs, numerical techniques have been developed over the last few decades and one of the prominent ones, today, is the Finite Element Analysis.

Differential equations not only describe natural phenomena but also physical phenomena encountered in engineering mechanics. These partial differential equations (PDEs) are complicated equations that need to be solved in order to compute relevant quantities of a structure (like stresses (ε), strains (ε), etc.) in order to estimate the structural behavior under a given load. It is important to know that FEA only gives an approximate solution to the problem and is a numerical approach to get the real result of these partial differential equations. Simplified, FEA is a numerical method used for the prediction of how a part or assembly behaves under given conditions. It is used as the basis for modern simulation software and helps engineers to find weak spots, areas of tension, etc. in their designs. The results of a simulation-based on the FEA method are usually depicted via a color scale that shows, for example, the pressure distribution over the object.

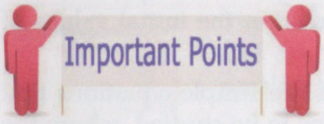
Harmonic response analysis gives the ability to predict the sustained dynamic behavior of structures, thus it enabling to verify whether or not designs will successfully overcome resonance, fatigue and other harmful effects of forced vibrations.

Simple harmonic motion is the movement of a simple harmonic oscillator; the action is intermittent, as it repeats itself at regular intervals in a definite manner-described as being sinusoidal, with regular amplitude. The harmonic analysis is carried out by means of MSC Nastran which is described by its amplitude, its period which is the time for a single fluctuation, its frequency which is the number of cycles per unit time, and its phase, which decided the initial position on the sine wave. In words simple harmonic motion is "movement where the acceleration of a body is relative to, and reverses in direction to the displacement from its equilibrium position". Simple harmonic motion can provide as a numerical form of a variety of motions and provides the foundation of the description of more difficult motions through the techniques of Fourier analysis.

2. DERIVATION OF THE ELEMENT STIFFNESS MATRIX

The finite element method (FEM) is a widely used method for numerically solving differential equations arising in engineering and mathematical modeling. Typical problem areas of interest include the traditional fields of structural analysis, heat transfer, fluid flow, mass transport, and electromagnetic potential. The FEM is a particular numerical method for solving partial differential equations in two or three space variables (i.e., some boundary value problems). To solve a problem, the FEM subdivides a large system into smaller, simpler parts that are called finite elements. This is achieved by a particular space discretization in the space dimensions, which is implemented by the construction of a mesh of the object: the numerical domain for the solution, which has a finite number of points.

The finite element method formulation of a boundary value problem finally results in a system of algebraic equations. The method approximates the unknown function over the domain. The simple equations that model these finite elements are then assembled into a larger system of equations that models the entire problem. The FEM then uses variational methods from the calculus of variations to approximate a solution by minimizing an associated error function. Studying or analyzing a phenomenon with FEM is often referred to as finite element analysis (FEA).



Typical problem areas of interest include the traditional fields of structural analysis, heat transfer, fluid flow, mass transport, and electromagnetic potential.

From inspection, we can see that there are two degrees of freedom in this model, u_i and u_j. We can write the force equilibrium equations:

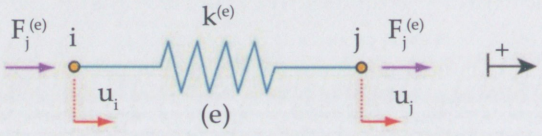


Fig. 3.1 Model with two degrees of freedom.

$$k^{(e)}u_i - k^{(e)}u_j = F_i^{(e)}$$

$$-k^{(e)}u_i + k^{(e)}u_j = F_j^{(e)}$$

In matrix form

$$\begin{bmatrix} k^e & -k^e \\ -k^e & k^e \end{bmatrix} \begin{Bmatrix} u^i \\ u^j \end{Bmatrix} = \begin{Bmatrix} k_i^{(e)} \\ k_j^{(e)} \end{Bmatrix}$$

The order of the matrix is [2×2] because there are 2 degrees of freedom. Note also that the matrix is symmetrical. The 'element' stiffness relation is:

$$[k^{(e)}]\ \{u^{(e)}\} = \{f^{(e)}\}$$

Where $K^{(e)}$ is the element stiffness matrix, $u^{(e)}$ the nodal displacement vector and $F^{(e)}$ the nodal force vector. (The element stiffness relation is important because it can be used as a building block for more complex systems. An example of this is provided later.)

Basic Concepts

The subdivision of a whole domain into simpler parts has several advantages:

- Accurate representation of complex geometry
- Inclusion of dissimilar material properties
- Easy representation of the total solution
- Capture of local effects.

Typical work out of the method involves (1) dividing the domain of the problem into a collection of subdomains, with each subdomain represented by a set of element equations to the original problem, followed by (2) systematically recombining all sets of element equations into a global system of equations for the final calculation. The global system of equations has known solution techniques, and can be calculated from the initial values of the original problem to obtain a numerical answer.

In the first step above, the element equations are simple equations that locally approximate the original complex equations to be studied, where the

original equations are often partial differential equations (PDE). To explain the approximation in this process, the Finite element method is commonly introduced as a special case of Galerkin method. The process, in mathematical language, is to construct an integral of the inner product of the residual and the weight functions and set the integral to zero. In simple terms, it is a procedure that minimizes the error of approximation by fitting trial functions into the PDE. The residual is the error caused by the trial functions, and the weight functions are polynomial approximation functions that project the residual. The process eliminates all the spatial derivatives from the PDE, thus approximating the PDE locally with

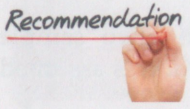
- A set of algebraic equations for steady state problems,
- A set of ordinary differential equations for transient problems.

These equation sets are the element equations. They are linear if the underlying PDE is linear, and vice versa. Algebraic equation sets that arise in the steady-state problems are solved using numerical linear algebra methods, while ordinary differential equation sets that arise in the transient problems are solved by numerical integration using standard techniques such as Euler's method or the Runge-Kutta method.

In step (2) above, a global system of equations is generated from the element equations through a transformation of coordinates from the subdomains' local nodes to the domain's global nodes. This spatial transformation includes appropriate orientation adjustments as applied in relation to the reference coordinate system. The process is often carried out by FEM software using coordinate data generated from the subdomains.

FEM is best understood from its practical application, known as finite element analysis (FEA). FEA as applied in engineering is a computational tool for performing engineering analysis. It includes the use of mesh generation techniques for dividing a complex problem into small elements, as well as the use of software program coded with FEM algorithm. In applying FEA, the complex problem is usually a physical system with the underlying physics such as the Euler-Bernoulli beam equation, the heat equation, or the Navier-Stokes equations expressed in either PDE or integral equations, while the divided small elements of the complex problem represent different areas in the physical system.

FEA is a good choice for analyzing problems over complicated domains (like cars and oil pipelines), when the domain changes (as during a solid-state reaction with a moving boundary), when the desired precision varies over the



The global system of equations has known solution techniques, and can be calculated from the initial values of the original problem to obtain a numerical answer.

entire domain, or when the solution lacks smoothness. FEA simulations provide a valuable resource as they remove multiple instances of creation and testing of hard prototypes for various high fidelity situations. For instance, in a frontal crash simulation it is possible to increase prediction accuracy in "important" areas like the front of the car and reduce it in its rear (thus reducing the cost of the simulation). Another example would be in numerical weather prediction, where it is more important to have accurate predictions over developing highly nonlinear phenomena (such as tropical cyclones in the atmosphere, or eddies in the ocean) rather than relatively calm areas.

The Structure of Finite Element Methods

A finite element method is characterized by a variational formulation, a discretization strategy, one or more solution algorithms, and post-processing procedures. Examples of the variational formulation are the Galerkin method, the discontinuous Galerkin method, mixed methods, etc.

A discretization strategy is understood to mean a clearly defined set of procedures that cover (a) the creation of finite element meshes, (b) the definition of basis function on reference elements (also called shape functions) and (c) the mapping of reference elements onto the elements of the mesh. Examples of discretization strategies are the h-version, p-version, hp-version, x-FEM, isogeometric analysis, etc. Each discretization strategy has certain advantages and disadvantages. A reasonable criterion in selecting a discretization strategy is to realize nearly optimal performance for the broadest set of mathematical models in a particular model class.

Various numerical solution algorithms can be classified into two broad categories; direct and iterative solvers. These algorithms are designed to exploit the sparsity of matrices that depend on the choices of variational formulation and discretization strategy.

Postprocessing procedures are designed for the extraction of the data of interest from a finite element solution. In order to meet the requirements of solution verification, postprocessors need to provide for a posteriori error estimation in terms of the quantities of interest. When the errors of approximation are larger than what is considered acceptable then the discretization has to be changed either by an automated adaptive process or by the action of the analyst. There are some very efficient postprocessors that provide for the realization of superconvergence.

Illustrative Problems P1 and P2

We will demonstrate the finite element method using two sample problems from which the general method can be extrapolated. It is assumed that the reader is familiar with calculus and linear algebra.

P1 is a one-dimensional problem

$$P1 : \begin{cases} u''(x) = f(x) \text{ in } (0,1), \\ u(0) = u(1) = 0, \end{cases}$$

where f is given, u is an unknown function of x, and u″ is the second derivative of u with respect to x.

P2 is a two-dimensional problem (Dirichlet problem)

$$P2 : \begin{cases} u_{xx}(x,y) + u_{yy}(x,y) = f(x,y) & \text{in } \Omega, \\ u = 0 & \text{on } \partial\Omega, \end{cases}$$

where Ω is a connected open region in the (x, y) plane whose boundary $\partial\Omega$ is nice (e.g., a smooth manifold or a polygon), and u_{xx} and u_{yy} denote the second derivatives with respect to x and y, respectively.

The problem P1 can be solved directly by computing antiderivatives. However, this method of solving the boundary value problem (BVP) works only when there is one spatial dimension and does not generalize to higher-dimensional problems or problems like u + u″ = f. For this reason, we will develop the finite element method for P1 and outline its generalization to P2.

Our explanation will proceed in two steps, which mirror two essential steps one must take to solve a boundary value problem (BVP) using the FEM.

- In the first step, one rephrases the original BVP in its weak form. Little to no computation is usually required for this step. The transformation is done by hand on paper.
- The second step is the discretization, where the weak form is discretized in a finite-dimensional space.

After this second step, we have concrete formulae for a large but finite-dimensional linear problem whose solution will approximately solve the original BVP. This finite-dimensional problem is then implemented on a computer.

Weak Formulation

The first step is to convert P1 and P2 into their equivalent weak formulations.

IMPORTANT

Various numerical solution algorithms can be classified into two broad categories; direct and iterative solvers.

The Weak Form of P1

If u solves P1, then for any smooth function v that satisfies the displacement boundary conditions, i.e. v = 0 at x = 0 and x = 1, we have

$$\int_0^1 f(x)v(x)dx = \int_0^1 u''(x)v(x)dx.$$

Conversely, if u with u(0) = u(1) = 0 satisfies (1) for every smooth function v (x) then one may show that this u will solve P1. The proof is easier for twice continuously differentiable u (mean value theorem), but may be proved in a distributional sense as well.

We define a new operator or map ϕ (u, v) by using integration by parts on the right-hand-side of (1):

$$\begin{aligned}\int_0^1 f(x)v(x)dx &= \int_0^1 u''(x)v(x)dx \\ &= u'(x)v(x)\,|_0^1 - \int_0^1 u'(x)v'(x)dx \\ &= -\int_0^1 u'(x)v'(x)dx \equiv -\phi(u,v),\end{aligned}$$

where we have used the assumption that v (0) = v(1) = 0.

The weak form of P2

If we integrate by parts using a form of Green's identities, we see that if u solves P2, then we may define ϕ (u, v) for any v by

$$\int_\Omega fvds = -\int_\Omega \nabla u \cdot \nabla v ds \equiv -\phi(u,v),$$

where ∇ denotes the gradient and $\cdot$ denotes the dot product in the two-dimensional plane. Once more ϕ can be turned into an inner product on a suitable space $H^1_0(\Omega)$ of once differentiable functions of Ω that are zero on $\partial\Omega$. We have also assumed that $v \in H^1_0(\Omega)$. Existence and uniqueness of the solution can also be shown.

A Proof Outline of Existence and Uniqueness of the Solution

We can loosely think of $H^1_0(0,1)$ to be the absolutely continuous functions of (0,1) that are 0 at x = 0 and x = 1. Such functions are (weakly) once differentiable and it turns out that the symmetric bilinear map ϕ then defines an inner product which turns $H^1_0(0,1)$ into a Hilbert space (a detailed proof is nontrivial). On the other hand, the left-hand-side $\int_0^1 (x)v(x)ds$ is also an inner product, this time on

the Lp space$L^2(0,1)$. An application of the Riesz representation theorem for Hilbert spaces shows that there is a unique u solving (2) and therefore P1. This solution is a-priori only a member of $H^1_0(0,1)$, but using elliptic regularity, will be smooth if f is.

3. ASSEMBLY OF THE OVERALL STIFFNESS MATRIX

Although, it isn't apparent for the simple two-spring model above, generating the global stiffness matrix (directly) for a complex system of springs is impractical. A more efficient method involves the assembly of the individual element stiffness matrices. For instance, if you take the 2-element spring system shown,

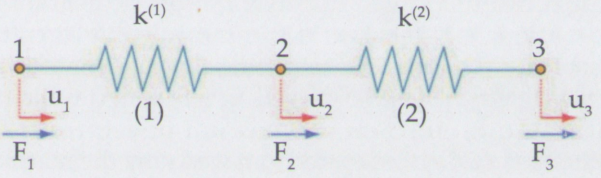


Fig. 3.2 2-Element spring system

split it into its component parts in the following way

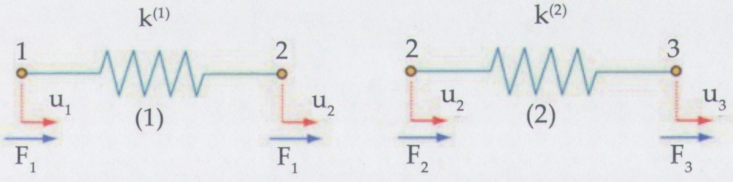


Fig. 3.3 Split of component

and derive the force equilibrium equations

$$k^1u_1 - k^1u_2 = F_1$$
$$k^1u_2 - k^1u_1 = k^2u_2 - k^2u_3 = F_2$$
$$k^2u_3 - k^2u_2 = F_3$$

then the individual element stiffness matrices are:

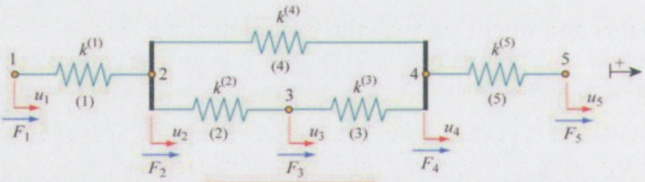
$$\begin{bmatrix} k^1 & -k^1 \\ -k^1 & k^1 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \end{Bmatrix} \quad \text{and} \quad \begin{bmatrix} k^2 & -k^2 \\ -k^2 & k^2 \end{bmatrix} \begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} F_2 \\ F_3 \end{Bmatrix}$$

such that the global stiffness matrix is the same as that derived directly:

$$\begin{bmatrix} k^1 & -k^1 & 0 \\ -k^1 & k^1+k^2 & -k^2 \\ 0 & -k^2 & k^2 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \\ F_3 \end{Bmatrix}$$

(Note that, to create the global stiffness matrix by assembling the element stiffness matrices, k^{22} is given by the sum of the direct stiffnesses acting on node 2 – which is the compatibility criterion. Note also that the indirect cells k_{ij} are either zero (no load transfer between nodes i and j), or negative to indicate a reaction force.)

For this simple case the benefits of assembling the element stiffness matrices (as opposed to deriving the global stiffness matrix directly) aren't immediately obvious. We consider therefore the following (more complex) system which contains 5 springs (elements) and 5 degrees of freedom (problems of practical interest can have tens or hundreds of thousands of degrees of freedom (and more!)). Since there are 5 degrees of freedom we know the matrix order is 5×5. We also know that it's symmetrical, so it takes the form shown below:



$$[K]\{u\} = \{F\}$$

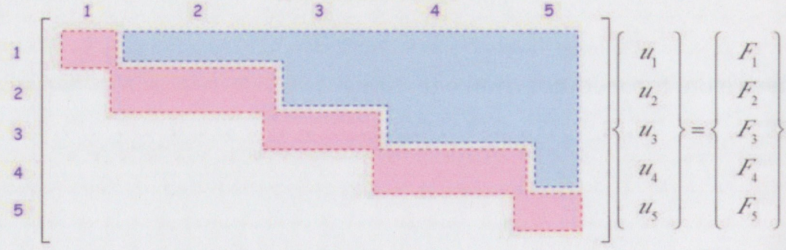


Fig. 3.4 Symmetrical form

We want to populate the cells to generate the global stiffness matrix. From our observation of simpler systems, e.g. the two spring system above, the following rules emerge:

- The term in location ii consists of the sum of the direct stiffnesses of all the elements meeting at node i

- The term in location ij consists of the sum of the indirect stiffnesses relating to nodes i and j of all the elements joining node i to j
- Add a negative for reaction terms ($-k_{ij}$)
- Add a zero for node combinations that don't interact

By following these rules, we can generate the global stiffness matrix:

$$\begin{array}{c} \\ 1 \\ 2 \\ 3 \\ 4 \\ 5 \end{array}\begin{array}{c} \begin{array}{ccccc} 1 & 2 & 3 & 4 & 5 \end{array} \\ \begin{bmatrix} k^{(1)} & -k^{(1)} & 0 & 0 & 0 \\ -k^{(1)} & k^{(1)}+k^{(2)}+k^{(4)} & -k^{(2)} & -k^{(4)} & 0 \\ 0 & -k^{(2)} & k^{(2)}+k^{(3)} & -k^{(3)} & 0 \\ 0 & -k^{(4)} & -k^{(3)} & k^{(3)}+k^{(4)}+k^{(5)} & -k^{(5)} \\ 0 & 0 & 0 & -k^{(5)} & k^{(5)} \end{bmatrix} \end{array} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \\ u_5 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \\ F_3 \\ F_4 \\ F_5 \end{Bmatrix}$$

Fig. 3.5 Global stiffness matrix

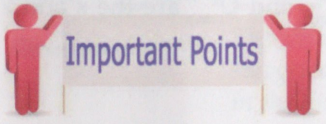
This type of assembly process is handled automatically by commercial FEM codes

Drag the springs into position and click 'Build matrix', then apply a force to node 5. You will then see the force equilibrium equations, the equivalent spring stiffness and the displacement at node 5.

4. COMPARISON WITH THE FINITE DIFFERENCE APPROACH

A paper which purports to make a comparison between finite difference and finite element methods is doomed to be met with much criticism. The reason for this inevitable fate is that a study such as this must contain a high degree of subjectivity which many readers will have cause to disagree with. Issues such as ulterior motives of the author, the proficiency of a computer programmer, the blurring of features which distinguish finite element from finite difference methods and the reader's own biases or preferences make presentation of a satisfactorily complete and inoffensive comparison of these two numerical techniques virtually impossible.

There are many possible ways to discretize the wave equation using the FDM; for example, the classical standard-grid displacement formulation (Kelly et al.1976), the staggered-grid velocity-stress formulation, the rotated staggered-grid formulation (Saenger et al.2000), the standard-grid velocity-stress formulation (VSFDM, Lombard et al.2008) and the implicit staggered-



The reason for this inevitable fate is that a study such as this must contain a high degree of subjectivity which many readers will have cause to disagree with.

grid formulation. It is beyond the scope of this paper to carry out an exhaustive comparison of all these formulations. Among these, we wish to focus on VSFDM, which has attracted relatively little attention despite its simplicity and accuracy. For comparison purposes, we will also give an overview of DFDM and SGFDM, which are related to VSFDM in ways that will become evident in the following subsections.

Standard-Grid Displacement Formulation

The first attempt to generate synthetic seismograms using FDM was carried out by Alterman & Karal (1968). They used the displacement formulation of the wave equation in polar coordinates and discretized it using the classical second-order finite-difference operator. This approach was later adapted by Alford et al. (1974) to the acoustic wave equation and by Kelly et al. (1976) to the elastic wave equation. It is interesting to note that, even though the above mentioned papers present a straightforward implementation of FDM, the formulation for the elastic case is not unique because there are different ways to discretize the cross-derivative term (the second term in the right-hand side in equation for $i \neq j$).

Since the comparison of the different methods will be based on the spatial discretization, we will only include the semidiscrete form of the equations. The discretization of the elastic wave equation by DFDM is given by:

$$\begin{aligned}
\partial^2 U^x_{m,n} &= \alpha^2 h^{-2}(U^x_{m+1,n} - 2U^x_{m,n} + U^x_{m-1,n}) \\
&\quad + \beta^2 h^{-2}(U^x_{m,n+1} - 2U^x_{m,n} + U^x_{m,n-1}) \\
&\quad + \frac{1}{4} h^{-2}(\alpha^2 - \beta^2)(U^z_{m,-1,n+1} - U^z_{m,+1,n-1} \\
&\quad - U^z_{m,-1,n+1} + U^z_{m,-1,n-1}), \\
\partial^2_t U^z_{m,n} &= \alpha^2 h^{-2}(U^z_{m+n|1} - 2U^z_{m,n} + U^z_{m,n\,1}) \\
&\quad + \beta^2 h^{-2}(U^z_{m+1,n} - 2U^z_{m,n} + U^z_{m-1,n}) \\
&\quad + \frac{1}{4} h^{-2}(\alpha^2 - \beta^2)(U^z_{m,+1,n+1} - U^x_{m,+1,n-1} \\
&\quad - U^x_{m,-1,n+1} + U^x_{m,-1,n-1}),
\end{aligned}$$

where h is the spatial increment in the x and z directions, $\hat{a} = \sqrt{(\ddot{e} + 2\dot{i})/\tilde{n}}$ and $\hat{a} = \sqrt{\dot{i}/\tilde{n}}$ are the P- and S-wave velocities, and $U^x_{m,n}$ and $U^z_{m,n}$ are the x and z components of displacement at the point (x_m, z_n).

Staggered-Grid Velocity-Stress Formulation

The SGFDM stencil takes advantage of the structure of the velocity-stress formulation to evaluate the derivatives using a centred finite-difference operator and obtain second-order accuracy in space. The method was originally developed by Yee for Maxwell's equations and then used in seismic modelling by Madariaga. This method was successfully applied to elastic wave propagation by Virieux and has been extended to higher-order approximations in Levander and to 3-D in Graves and Minkoff. See also Dougherty & Stephen and Stephen for an application of this method to sea-surface scattering in 2-D and Burns & Stephen for an application to scattering in the presence of 3-D sea-surface topography.

The semidiscrete version of the staggered-grid equations is given by (Virieux)

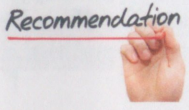
$$\partial_t V^x_{m+1/2,n} = \frac{\overline{b}_{m+1/2,n}}{h}(T^{xx}_{m+1,n} - T^{xx}_{m,n} + T^{xz}_{m+1/2,n+1/2} - T^{xz}_{m+1/2,n-1/2}),$$

$$\partial_t V^z_{m,n+1/2} = \frac{\overline{b}_{m,n+1/2}}{h}(T^{xz}_{m+1/2,n+1/2} - T^{xz}_{m-1/2,n+1/2} + T^{zz}_{m,n+1} - T^{zz}_{m,n}),$$

$$\partial_t T^{xx}_{m,n} = \frac{\lambda_{m,n} + 2\mu_{m,n}}{h}(V^x_{m+1/2,n} - V^x_{m-1/2,n}) + \frac{\lambda_{m,n}}{h}(V^z_{m,n+1/2} - V^z_{m,n-1/2}),$$

$$\partial_t T^{zz}_{m,n} = \frac{\lambda_{m,n}}{h}(V^x_{m+1/2,n} - V^x_{m-1/2,n}) + \frac{\lambda_{m,n} + 2\mu_{m,n}}{h}(V^z_{m,n+1/2} - V^z_{m,n-1/2}),$$

where $V^x_{m,n}$ and $V^z_{m,n}$ are the x and z components of the velocity field, and



The discretization in time is usually also implemented in a staggered fashion such that the method is second-order accurate in the time domain.

$T^{xx}_{m,n}$, $T^{zz}_{m,n}$ and $T^{xz}_{m,n}$ are the components of the stress tensor, $\overline{b}_{m+1/2,n}$ is the average buoyancy between nodes (m, n) and (m + 1, n), $\overline{b}_{m+1/2}$ is the average buoyancy between nodes (m, n) and (m, n + 1), and $\mu_{m+1/2+1/2}$ is the average of μ at the centre of the cell defined by the nodes (m, n), (m + 1, n), (m, n + 1) and (m + 1, n + 1). The averaging adopted in this paper is the one adopted by Graves (1996), which is given by

$$\overline{b}_{m,n+1/2} = \frac{1}{2}(\rho^{-1}_{m,n} + \rho^{-1}_{m,n+1}),$$

$$\overline{\mu}_{m+1/2,n+1/2} = 4(\mu^{-1}_{m,n} + \mu^{-1}_{m+1,n} + \mu^{-1}_{m,n+1} + \mu^{-1}_{m+1,n+1})^{-1}.$$

The discretization in time is usually also implemented in a staggered fashion such that the method is second-order accurate in the time domain. The above equations can be easily modified for higher-order differentiation, see for example Levander and Mittet. One of the advantages of this approach is that it effectively doubles the sampling ratio because of the staggering. Another advantage is that the grid dispersion is significantly improved with respect to DFDM (this property will be reviewed in the next section). The disadvantages are that the number of degrees of freedom is larger than that of DFDM and that it is impossible to match the medium discontinuities with a staggered grid, even for simple models with straight interfaces, and therefore the error associated with the misalignment is unavoidable. Furthermore, Lombard et al. (2008) summarized the difficulties and accuracy limitations of imposing free-surface boundary conditions and proposed an alternative strategy that is incompatible with a staggered grid.

5. VARIATIONAL FORMULATION

In this section it will discuss the variational formulation for boundary integral equations and its connection to the variational solution of partial differential equations. We collect some basic theorems in functional analysis which are needed for this purpose. In particular, Green's theorems and the Lax–Milgram theorem are fundamental tools for the solvability of boundary integral equations as well as for elliptic partial differential equations. We will present here a subclass of boundary value problems for which the coerciveness property for some associated boundary integral operators follows directly from that of the variational form of the boundary and transmission problems. In this class, the solution of the boundary integral equations will be established with the help of the existence and regularity results of elliptic partial differential equations in variational form.

The following sections derive variational formulations for some prototype differential equations in 1D, and demonstrate how we with ease can handle variable coefficients, mixed Dirichlet and Neumann boundary conditions, first-order derivatives, and nonlinearities.

Variable Coefficient

Consider the problem

$$-\frac{\mathrm{d}}{\mathrm{dx}}\left(\alpha(\mathrm{x})\frac{du}{dx}\right)=\mathrm{f}(\mathrm{x}),\ \mathrm{x}\in\Omega=[0,\mathrm{L}],\mu(0)=\mathrm{C},\mathrm{u}(\mathrm{L})=\mathrm{D}.$$

There are two new features of this problem compared with previous examples: a variable coefficient $\alpha(x)$ and nonzero Dirichlet conditions at both boundary points.

Let us first deal with the boundary conditions. We seek

$$u(\mathrm{x})=\mathrm{B}(\mathrm{x})+\sum_{j\in\mathrm{I}_s}c_j\psi_i(\mathrm{x}).$$

Since the Dirichlet conditions demand

$\psi_i(0) = \psi_i(L) = 0$ i$\in I_s$,

$$\phi_i(0)=\phi_i(L)=0,\quad i\in\mathcal{I}_s,$$

the function B(x) must fulfill B(0) = C and B(L) = D. The we are guaranteed that u(0) = C and u(L) = D. How B varies in between x = 0 and x = L is not of importance. One possible choice is

$$B(x)=C+\frac{1}{L}(D-C)x,$$

which follows from (48) with p = 1.

We seek (u–B)$\in$ v. As usual,

$$\mathrm{V}=\mathrm{span}\ \{\Psi_0,\ldots,\Psi_\mathrm{N}\}$$

Note that any v $\in$ v has the property v(0) = v(L) = 0.

The residual arises by inserting our u in the differential equation:

$$\mathrm{R}=-\frac{d}{dx}\left(\alpha\frac{du}{dx}\right)-\mathrm{f}.$$

Galerkin's method is

$$(R,v)=0,\quad \forall v\in V,$$

or written with explicit integrals,

$$\int_{\Omega}(-\frac{d}{dx}(\acute{a}\frac{du}{dx})-f)v\mathrm{d}x=0,\quad \forall v\in V.$$

We proceed with integration by parts to lower the derivative from second to first order:

IMPORTANT

The solution of the boundary integral equations will be established with the help of the existence and regularity results of elliptic partial differential equations in variational form.

$$-\int_{\hat{U}} \frac{d}{dx}(\acute{a}(x)\frac{du}{dx})v\,dx = \int_{\hat{U}} \acute{a}(x)\frac{du}{dx}\frac{dv}{dx}dx - [\acute{a}\frac{du}{dx}v]_0^L.$$

The boundary term vanishes since v(0) = v(L) = 0. The variational formulation is then

$$\int_{\Omega} \acute{a}(x)\frac{du}{dx}\frac{dv}{dx}dx = \int_{\Omega} f(x)v\,dx, \quad \forall v \in V.$$

The variational formulation can alternatively be written in a more compact form:

$$(\acute{a}u', v') = (f, v), \quad \forall v \in V.$$

The corresponding abstract notation reads

$$a(u, v) = L(v) \quad \forall v \in V,$$

with

$$\alpha(u,v) = (\alpha u', v'), \; L(v) = (f,v).$$

We may insert $u = B + \sum_j c_j \psi_j$ and $v = \Psi_i$ to derive the linear system:

$$(\alpha B' + \alpha \sum_{j \in \mathcal{I}_s} c_j \psi_j', \psi_i') = (f, \psi_i), \quad i \in \mathcal{I}_s.$$

Isolating everything with the c_j coefficients on the left-hand side and all known terms on the right-hand side gives

$$\sum_{j \in \mathcal{I}_s} (\alpha \psi_j', \psi_i') c_j = (f, \psi_i) - (\alpha(D-C)L^{-1}, \psi_i'), \quad i \in \mathcal{I}_s.$$

This is nothing but a linear system $\sum_j A_{i,j} c_j = b_i$ with

$$A_{i,j} = (\alpha \psi_j', \psi_i') = \int_{\Omega} \alpha(x)\psi_j'(x), \psi_i'(x)\,dx,$$

$$b_i = (f, \psi_i) - (\alpha(D-C)L^{-1}, \psi_i') = \int_{\Omega} (f(x)\psi_i(x) - \alpha(x)\frac{D-C}{L}\psi_i'(x))dx.$$

6. BOUNDARY CONDITIONS

Boundary conditions (B.C.) are constraints necessary for the solution of a boundary value problem. A boundary value problem is a differential equation (or system of differential equations) to be solved in a domain on whose boundary a set of conditions is known. It is opposed to the "initial value problem", in which only the conditions on one extreme of the interval are known. Boundary value problems are extremely important as they model a vast amount of phenomena and applications, from solid mechanics to heat transfer, from fluid mechanics to acoustic diffusion.

They arise naturally in every problem based on a differential equation to be solved in space, while initial value problems usually refer to problems to be solved in time. Boundary value problems have been extensively studied by Jacques Charles François Sturm and Joseph Liouville, who studied the eigenvalues of a linear differential equation of the second order. They studied the conditions which guarantee the existence and uniqueness of the solution of the differential problem and how it is affected by the boundary conditions2. The Sturm-Liouville theory is extremely important for any computational problem because it enables to understand if a problem is "well-posed" and how it is possible to obtain the solution.

Types of Boundary Conditions

Both ordinary and partial differential equations need boundary conditions to be solved. Different types of boundary conditions can be imposed on the boundary of the domain. The choice of the boundary condition is fundamental for the resolution of the computational problem: a bad imposition of B.C. may lead to the divergence of the solution or to the convergence to a wrong solution.

Dirichlet Boundary Condition

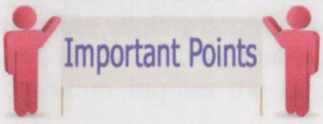
The Dirichlet boundary condition is a type of boundary condition named after Peter Gustav Lejeune Dirichlet (1805–1859).

This condition specifies the value that the unknown function needs to take on along the boundary of the domain. Given, for example, the Laplace equation, the boundary value problem with the Dirichlet b.c. is written as:

$$\Delta\varphi(\underline{x}) = 0 \qquad \forall \underline{x} \in \Omega$$

$$\varphi(\underline{x}) = f(\underline{x}) \qquad \forall \underline{x} \in \partial\Omega$$

where φ is the unknown function, $\underline{x}$ is the independent variable (e.g. the spatial coordinates), Ω is the function domain, $\partial\Omega$ is the boundary of the domain, and f is a given scalar function defined on $\partial\Omega$. In the framework of numerical simulations, it is usually imposed directly in the algebraic system to be solved. Let's consider the following algebraic system derived from a numerical algorithm:



Boundary value problems have been extensively studied by Jacques Charles François Sturm and Joseph Liouville, who studied the eigenvalues of a linear differential equation of the second order

$$\begin{bmatrix} K_{1,1} & K_{2,2} & \cdot & K_{1,M-1} & K_{1,M} \\ K_{2,1} & K_{2,2} & \cdot & K_{2,M-1} & K_{2,M} \\ \cdot & \cdot & \cdot & \cdot & \cdot \\ K_{M-1,1} & K_{M-1,2} & \cdot & K_{M-1,\,M-1} & K_{M-1M} \\ K_{M,1} & K_{M,2} & \cdot & K_{M,M-1} & K_{M,M} \end{bmatrix} \begin{bmatrix} X_1 \\ X_2 \\ \cdot \\ X_{N-1} \\ X_N \end{bmatrix} = \begin{bmatrix} A_1 \\ A_2 \\ \cdot \\ A_{M-1} \\ A_M \end{bmatrix}$$

where k_{ij} are the elements of the algebraic operator (e.g. the stiffness matrix), x_i are the unknowns (i.e. the degrees of freedom of the problem), and a_i are the known terms. The simplest way to impose a Dirichlet boundary condition on the n^{th} degree of freedom is to modify the system as follows:

$$\begin{bmatrix} k_{1,1} & \cdot & \cdot & \cdot & k_{1,m} \\ \cdot & \cdot & \cdot & \cdot & \cdot \\ 0 & 0 & 1 & 0 & 0 \\ \cdot & \cdot & \cdot & \cdot & \cdot \\ k_{m,1} & \cdot & \cdot & \cdot & k_{m,m} \end{bmatrix} \begin{bmatrix} x_1 \\ \cdot \\ x_n \\ \cdot \\ x_m \end{bmatrix} = \begin{bmatrix} a_1 \\ \cdot \\ f \\ \cdot \\ a_m \end{bmatrix}$$

where f is the value that the nth degree of freedom must be taking.

Neumann Boundary Condition

The Neumann boundary condition is a type of boundary condition, named after Carl Neumann. When imposed on an ordinary (ODE) or a partial differential equation (PDE), it specifies the values that the derivative of a solution is going to take on the boundary of the domain. Given, for example, the Laplace equation, the boundary value problem with the Neumann b.c. is written as:

$$\Delta\varphi(\underline{x}) = 0 \qquad \forall \underline{x} \in \Omega$$

$$\frac{\partial\varphi(\underline{x})}{\partial n} = f(\underline{x}) \qquad \forall \underline{x} \in \partial\Omega$$

where n is the unit normal to the boundary surface, if $\Omega \subset R^3$.

In the case of ODE (i.e. $\Omega \subset R^1$), the derivative normal to the boundary coincides with the global derivative φ'. In the rare cases in which a temporal dependency is solved through a finite element approach (instead of the more usual finite difference), this type of boundary condition is the most common. Neumann boundary condition is also called "natural" because it naturally appears in the development of the weak formulation in any finite element approach. Let's consider the following simple equation:

$$-u''(x) = p(x) \qquad \forall x \in R$$

where u is an unknown scalar field and p is a given scalar function. This equation rules many phenomena, for instance, the thermal diffusion in 1D and the tension/compression of a beam. The finite element method consists in rewriting the equation from a differential (strong) form to an integral (weak) formulation. This transformation is done through two steps:

1: Test and integrate:

$$-\int_a^b u''(x)v(x)dx = \int_a^b p(x)v(x)dx$$

where $v(x)$ is the shape function.

2: Apply the Green theorem to have a uniform distribution of derivatives and avoid higher order derivatives:

$$\int_a^b u'(x)v'(x)dx = \int_a^b p(x)v(x)dx + [u'(x)v(x)]_a^b$$

Thus, a term including the derivative of the unknown field on the boundary naturally appears; for 1D problems, this term refers to the extremes of the interval, for 2D problems it refers to the contour of the domain, and for 3D problems it refers to the boundary surfaces. The presence of the boundary term on the right-hand side highlights two properties of Neumann boundary conditions:

- homogeneous Neumann b.c. are naturally satisfied without any explicit imposition
- since Dirichlet b.c. are usually applied by modifying the right-hand term, a homogeneous Neumann condition is applied on all the boundaries where a Dirichlet b.c. is imposed.

Mixed Boundary Condition

It consists of applying different types of boundary conditions in different parts of the domain. It is important to notice that boundary conditions must be applied on the whole boundary: the "free" boundary is anyways subjected to a homogeneous Neumann condition. The mixed boundary condition differs from the Robin condition because the latter consists of different types of boundary conditions applied to the same region of the boundary, while the mixed condition implies different types of b.c. applied to different parts of the boundary.

Cauchy Boundary Condition

The Cauchy boundary condition is a condition on both the unknown field and its derivatives. It differs from the Robin condition because the Cauchy condition implies the imposition of two constraints (1 Dirichlet b.c. + 1

Neumann b.c.), while the Robin condition implies only one constraint on the linear combination of the unknown function and its derivatives.

Applications

Structural and Solid mechanics

Dirichlet boundary conditions: Solid mechanics is usually modeled through a displacement-based model, thus Dirichlet boundary conditions usually consist of imposing the displacement of the structure in given points. Structural mechanics is often based on formulations which include relative rotations, whose numerical resolution requires nonlinear shape functions in the finite element approximation. For instance, frame structures are based on the beam theory and the related finite element has 6 degrees of freedom (3 displacements + 3 rotations in a 3D space). For a 2D problem, each node of the boundary has 3 degrees of freedom on which Dirichlet boundary conditions can be applied: 2 displacements (ux and uy) and 1 rotation (ω).

Neumann boundary conditions

In solid mechanics, the spatial derivatives of displacements are related to the strain tensor. In elasticity, the strain is proportional to the stress, hence the Neumann boundary condition refers to both imposed strains and stresses. Since the stress is also linked to the external forces through the Cauchy's stress principle, the Neumann condition is also used to apply external loads. As stated in the section dedicated to Neumann boundary conditions, the homogeneous condition is naturally satisfied, so "free" boundaries may not be modeled explicitly.

Robin boundary conditions

It is used to model the mechanical impedance of a structure, that is how much it resists to motion when subjected to a harmonic load.

Fluid Mechanics

Dirichlet boundary conditions

In computational fluid mechanics, the classical Dirichlet boundary condition consists of the value of velocity and/or pressure to be taken by a certain set of nodes. It is common to refer to some sets of b.c. according to the following terminology:

- **Slip boundary condition:** the velocity normal to the boundary is set to zero, while the velocity parallel to the boundary is let free

- **No-slip boundary condition:** both the velocity normal to the boundary and the velocity parallel to the boundary are set equal to zero.

At least one homogeneous b.c. on the pressure (i.e. $p = 0$) has to be imposed as a reference for open domains, for instance, in the highest boundary of the air domain.

Neumann boundary conditions

Constraints on the derivative of velocity or pressure fields are mainly used in two cases. The first case is the application of a symmetry plane, thus:

$$\frac{\partial u}{\partial n} = 0$$

Since this condition is always applied in addition to a Dirichlet b.c. it is naturally satisfied. The second application is the modeling of wall friction, in the case when it is proportional to the strain rate:

$$S = (\nabla u + \nabla Tu)$$

Robin boundary conditions

It is used to describe semi-reflective walls, which partially absorb waves. It is not a very common application and it can be used only for pressure-based models. It is mostly used for acoustic applications.

7. SOLUTION OF THE LINEAR EQUATIONS

As discussed in previous topic of this unit, linear equation is a mathematical statement or equation that has only one variable in it. We know that to solve variables in equation number of equation should be equal to the number of the variables. So, to solve variable present in a linear equation of one variable, one equation is enough to solve the variable.

Below are some examples of linear equations in 1 variable, 2 variables and 3 variables:

Table 3.1 Examples of linear equations

Linear Equation in One variable	Linear Equation in Two variable	Linear Equation in Three variable
3x+5=0	y+7x=3	x + y + z = 0
32x+7=0	3a+2b = 5	a – 3b = c
98x = 49	6x+9y-12=0	3x + 12 y = ½ z

Below are given some examples of linear equation in one variable:

1. $2x + 3 = 35$
2. $3y + 34 = 8$

3. $2z + 15 = 89$

4. $18x + 45 = 23$

Above are the examples of linear equations in one variable.

Now following are the steps used in solving a linear equation in one variable:

- **Step I:** Observe the linear equation carefully.
- **Step II:** Carefully note the quantity you need to find out.
- **Step III:** Divide the equation in two parts, i.e., L.H.S. and R.H.S.
- **Step IV:** Figure out the terms containing constants and variables.
- **Step V:** Transfer all the constants on the Right Hand Side (R.H.S) of the equation and variables on the Left Hand Side (L.H.S.) of the equation.
- **Step VI:** Perform the algebraic operations on both sides of the equation to get the value of the variable.

Let us solve few examples to understand the concept in a better way.

1. Solve $x + 12 = 23$.

Solution: Let us first transfer the constants and variables on the R.H.S. and L.H.S. respectively. So,

$x = 23 - 12$

$x = 11.$

So, value of 'x' is 11.

2. Solve $2x + 13 = 43$.

Solution: Transfer the constants and variables on their respective sides. So,

$2x = 43 - 13$

$2x = 30$

$x = 30/2$

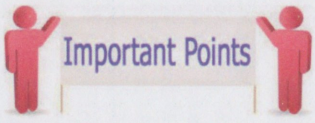
$x = 15.$

So, the value of 'x' is 15.

3. Solve $3x + 45 = 9x + 25$.

Solution: Transferring the variables and constants on the respective sides of the equation, we get,

$3x - 9x = 25 - 45$



To solve variable present in a linear equation of one variable, one equation is enough to solve the variable.

$-6x = -20$

$x = 20/6$

$x = 10/3.$

So, the value of the variable, $x = 10/3$.

Forming linear equations in one variable from given word problem and solving them:

Following are the steps involved in formation of linear equation from the given word problem:

Step I: First of all read the given problem carefully and note down the given and required quantities separately.

Step II: Denote the unknown quantities as 'x', 'y', 'z', etc.

Step III: Then translate the problem into mathematical language or statement.

Step IV: Form the linear equation in one variable using the given conditions in the problem.

Step V: Solve the equation for the unknown quantity.

Formulas

There are different forms to write linear equations. Some of them are:

Table 3.2 Different forms to write linear equations

Linear Equation	General Form	Example
Slope intercept form	$y = mx + c$	$y + 2x = 3$
Point–slope form	$y - y_1 = m(x - x_1)$	$y - 3 = 6(x - 2)$
General Form	$Ax + By + C = 0$	$2x + 3y - 6 = 0$
Intercept form	$x/x_0 + y/y_0 = 1$	$x/2 + y/3 = 1$
As a Function	f(x) instead of y	$f(x) = x + C$
	$f(x) = x + 3$	
The Identity Function	$f(x) = x$	$f(x) = 3x$
Constant Functions	$f(x) = C$	$f(x) = 6$

Standard Form of Linear Equation

Linear equations are a combination of constants and variables.

The standard form of a linear equation in one variable is represented as $ax + b = 0$ where, $a \neq 0$ and x is the variable.

The standard form of a linear equation in two variables is represented as $ax + by + c = 0$, where, $a \neq 0$, $b \neq 0$, x and y are the variables.

The standard form of a linear equation in three variables is represented as $ax + by + cz + d = 0$ where $a \neq 0$, $b \neq 0$, $c \neq 0$, x, y, z are the variables.

Slope Intercept Form

The most common form of linear equations is in slope-intercept form, which is represented as;

$$y = mx + c$$

where y and x are the point in x-y plane, m is the slope of the line (also called gradient) and c is the intercept (a constant value).

For example, $y = 3x + 7$:

$$\text{slope, } m = 3 \text{ and intercept} = 7$$

Point Slope Form

In this form of linear equation, a straight line equation is formed by considering the points in x-y plane, such that:

$$y - y_1 = m(x - x_1)$$

where (x_1, y_1) are the coordinates of the line.

We can also express it as:

$$y = mx + y_1 - mx_1$$

Intercept Form

A line which is neither parallel to x-axis or y-axis nor it pass through the origin but intersects the axes in two different points, represents the intercept form. The intercept values x_0 and y_0 of these two points are nonzero and forms an equation of the line as:

$$x/x_0 + y/y_0 = 1$$

Two-Point Form

If there are two points say, (x_1, y_1) and (x_2, y_2) and only one line passes through them, then the equation of the line is given by:

$$y - y_1 = [(y_2 - y_1)/(x_2 - x_1)](x - x_1)$$

where $(y_2 - y_1)/(x_2 - x_1)$ is the slope of the line and $x_1 \neq x_2$

Now Let us Try to form Some Linear Equations from Given Word Problems.

1. Sum of two numbers is 48. If one number is 5 times the other, find the numbers.

Solution: Let one of the numbers be 'x'. then second number is 5x.

Then, $x + 5x = 48$

$6x = 48$

$x = 48/6$

$x = 8.$

So 1st number = 8.

2nd number = $5x = 5 \times 8 = 40$.

2. A total of $ 34,000 is distributed as award prices among students. If the cash contains $ 100 and $ 500 noted in the ratio of 2 : 3. Then calculate the number of $ 100 and $ 500 notes that were distributed.

Solution: Since we are being given about the ratio of $ 100 as well as $ 500 notes.

So,

Let common ratio of number of notes be 'x'. Then,

Number of $ 100 notes = 2x.

Number of $ 500 notes = 3x.

Total amount = $100 \times 2x + 500 \times 3x$

$= 200x + 1500x$

$= 1700x$

Since total amount distributed is $ 14,000.

So, $1700x = 14,000$

$x = 14,000/1,700$

$x = 20.$

So, number of $ 100 notes = $2 \times 20 = 40$

Number of $ 500 notes = $3 \times 20 = 60$.

8. CONVERGENCE OF FINITE ELEMENT METHODS

Adaptive finite element methods (FEMs) have been widely used in applications for over 20 years now. In practice, they converge starting from coarse grids, although no mathematical theory has been able to prove this assertion. Ensuring an error reduction rate based on a posteriori error estimators, together with a reduction rate of data oscillation (information

missed by the underlying averaging process), we construct a simple and efficient adaptive FEM for elliptic partial differential equations. We prove that this algorithm converges with linear rate without any preliminary mesh adaptation nor explicit knowledge of constants.

Any prescribed error tolerance is thus achieved in a finite number of steps. A number of numerical experiments in two and three dimensions yield quasi-optimal meshes along with a competitive performance. Extensions to higher order elements and applications to saddle point problems are discussed as well.

The a priori convergence of finite element methods is based on the density property of the sequence of finite element spaces which essentially assumes a quasi-uniform mesh-refining. The advantage is guaranteed convergence for a large class of data and solutions; the disadvantage is a global mesh refinement everywhere accompanied by large computational costs. Adaptive finite element methods (AFEMs) automatically refine exclusively wherever their refinement indication suggests to do so and consequently leave out refinements at other locations. In other words, the density property is violated on purpose and the a priori convergence is not guaranteed automatically and, in fact, crucially depends on algorithmic details.

The advantage of AFEMs is a more effective mesh in many practical examples accompanied by smaller computational costs; the disadvantage is that the desirable convergence property is not guaranteed a priori. Efficient error estimators can justify a numerical approximation a posteriori and so achieve reliability. But it is not theoretically justified from the start that the adaptive mesh-refinement will generate an accurate solution at all. In order to foster the development of a convergence theory and improved design of AFEMs in computational engineering and sciences, this paper describes a particular version of an AFEM and analyses convergence results for three model problems in computational mechanics: linear elastic material (A), nonlinear monotone elastic material (B), and Hencky elastoplastic material (C). It establishes conditions sufficient for error-reduction in (A), for energy-reduction in (B), and eventually for strong convergence of the stress field in (C) in the presence of small hardening.

A typical engineering design involves the prediction of deflections/ displacements, stresses, natural frequencies, temperature distributions, etc.



Ensuring an error reduction rate based on a posteriori error estimators, together with a reduction rate of data oscillation, we construct a simple and efficient adaptive FEM for elliptic partial differential equations.

These parameters are used to iterate on material parameters and/or geometry to optimize their behavior. Traditional methods, like hand calculations, involved idealization of physical models using simple equations to obtain solutions. However, these approximations oversimplify the problem, and an analytical solution can only provide conservative estimates. Alternatively, FEM and other numerical methods are meant to provide an engineering analysis that takes into account much greater detail—something that would be impractical with hand calculations. FEM divides the body into smaller pieces, enforcing continuity of displacements along these element boundaries.

Convergence in Finite Element Analysis (FEA)

For those, using finite element analysis, the term "convergence" is often used. Most linear problems do not need an iterative solution procedure. Mesh convergence is an important issue that needs to be addressed. Additionally, in nonlinear problems, convergence in the iteration procedure also needs to be considered. So, what does this mean? In this article, we investigate and address issues related to this term.

Mesh Convergence: h- and p-refinement in Finite Element Analysis

One of the most overlooked issues in computational mechanics that affects accuracy is mesh convergence. This is related to how small the elements need to be to ensure that the results of the finite element analysis are not affected by changing the size of the mesh.

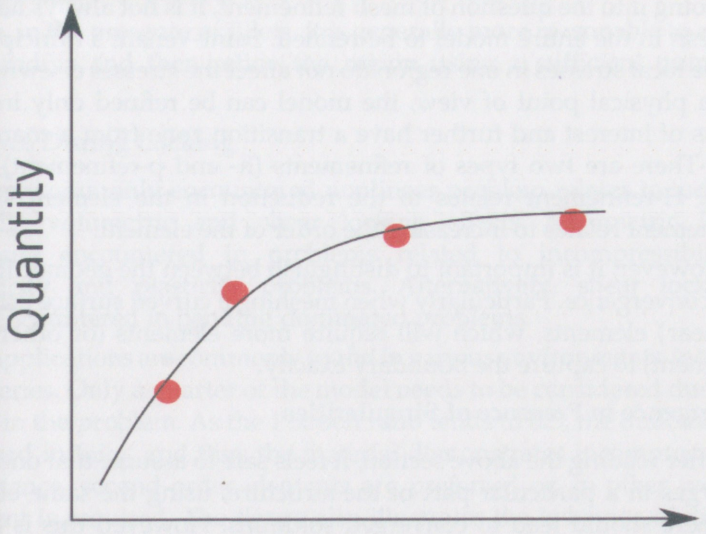


Fig. 3.6 Convergence of quantity with an increase in degrees of freedom

As shown in Fig. it is critical to first identify the quantity of interest. At least three points need to be considered, and as the mesh density increases, the quantity of interest starts to converge to a particular value. If two subsequent mesh refinements do not change the result substantially, then one can assume that the result has converged.

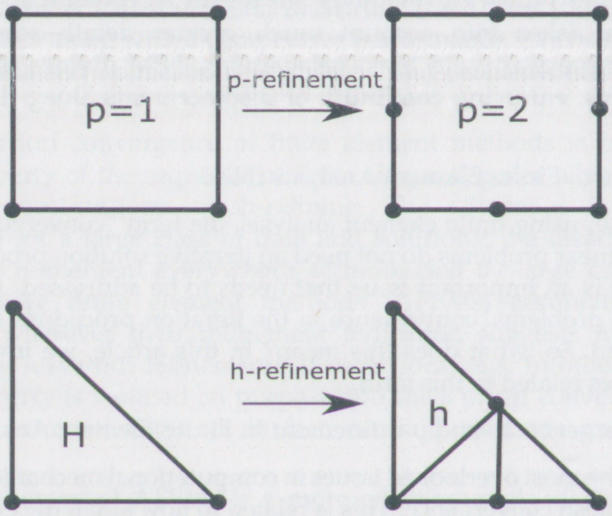


Fig. 3.7 Mesh refinement of a structure

Going into the question of mesh refinement, it is not always necessary for the mesh in the entire model to be refined. Saint-Venant's Principle enforces that the local stresses in one region do not affect the stresses elsewhere. Hence, from a physical point of view, the model can be refined only in particular regions of interest and further have a transition zone from a coarse to a fine mesh. There are two types of refinements (h- and p-refinement), as shown in Fig. H-refinement relates to the reduction in the element sizes, while p-refinement relates to increasing the order of the element.

However, it is important to distinguish between the geometric effect and mesh convergence. Particularly when meshing a curved surface using straight (or linear) elements, which will require more elements (or otherwise mesh refinement) to capture the boundary exactly.

Convergence in Presence of Singularities

After reading the above section, it feels safe to assume that once the stress converges in a particular part of the structure, using the same element size elsewhere should lead to converged solutions. However, this is not a valid assumption.

Most models have corners, both internal and external, where the radius is assumed to be zero. This is also the case in the presence of cracks. In these instances, the stresses are theoretically infinite. Now can you guess why airplane windows do not have corners but are rounded at the edges?

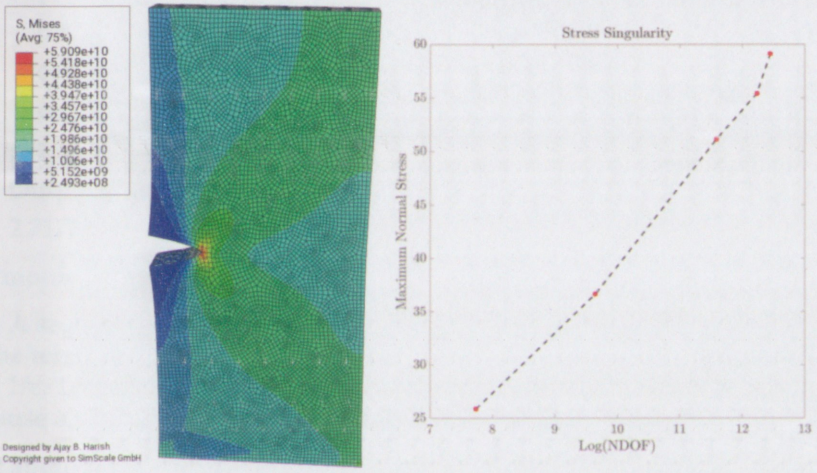


Fig. 3.8 Stress singularity

In the presence of the singularity, the mesh needs to be refined around it. However, as shown in Figure above, the more the mesh is refined, the more the stress continues to increase and tend towards infinity.

Hence, in the presence of fillets, it is generally more reasonable to assume an actual radius and then refine the region using a sufficient number of elements.

Convergence During Locking

Another commonly encountered nonlinear problem relates to locking—namely, the volumetric and shear locking effects. Volumetric locking is commonly encountered in problems related to incompressibility in hyperelasticity and plasticity problems. Alternatively, shear locking is generally encountered in bending dominated problems.

Such applications are commonly found in various environments, including human arteries. Only a quarter of the model needs to be considered due to the symmetry in the problem. As the Poisson ratio tends to 0.5, the bulk modulus tends toward infinity, and thus the material demonstrates incompressibility. In this instance, second-order elements are preferred or, in other words, a p-refinement is required. The figure also illustrates the behavior of different element types with an increase in the Poisson ratio.

9. A COMPUTER PROGRAM FOR HARMONIC PROBLEMS

Given a number N. The task is to find the Nth Harmonic Number.

Let the nth harmonic number be H_n.

The harmonic series is as follows:

$H_1 = 1$

$H_2 = H_1 + 1/2$

$H_3 = H_2 + 1/3$

$H_4 = H_3 + 1/4$

.

.

.

$H_n = H_{n-1} + 1/n$

Examples:

Input : N = 5

Output : 2.45

Input : N = 9

Output : 2.71786

The idea is to traverse from H_1 and then consecutively keep finding H_2 from H_1, H_3 from H_2 ..... and so on.

Below is the program to find N-th Harmonic Number:

```
// CPP program to find N-th Harmonic Number
#include <iostream>
using namespace std;
// Function to find N-th Harmonic Number
double nthHarmonic(int N)
{
  // H1 = 1
  float harmonic = 1.00;
  // loop to apply the forumula
  // Hn = H1 + H2 + H3 ... + Hn-1 + Hn-1 + 1/n
  for (int i = 2; i <= N; i++) {
    harmonic += (float)1 / i;
  }
   return harmonic;
```

```
}
// Driver Code
int main()
{
int N = 8;
cout<<nthHarmonic(N);
return 0;
}
```

Output:

2.71786

Harmonic Progression

A sequence of numbers is called a Harmonic progression if the reciprocal of the terms are in AP. In simple terms, a, b, c, d, e, f are in HP if 1/a, 1/b, 1/c, 1/d, 1/e, 1/f are in AP. For example, 1/a, 1/(a+d), 1/(a+2d), and so on are in HP because a, a + d, a + 2d are in AP.

Fact about Harmonic Progression

1. In order to solve a problem on Harmonic Progression, one should make the corresponding AP series and then solve the problem.
2. As the nth term of an A.P is given by an = a + (n-1)d, So the nth term of an H.P is given by 1/ [a + (n -1) d].
3. For two numbers, if A, G and H are respectively the arithmetic, geometric and harmonic means, then
   - $A \geq G \geq H$
   - $A\ H = G^2$, i.e., A, G, H are in GP
4. If we need to find three numbers in a H.P. then they should be assumed as 1/a–d, 1/a, 1/a+d
5. Majority of the questions of H.P. are solved by first converting them into A.P

How we check whether a series is harmonic progression or not?

The idea is to reciprocal the given array or series. After reciprocal, check if differences between consecutive elements are same or not. If all differences are same, Arithmetic Progression is possible. So as we know if the reciprocal of the terms are in AP then given a sequence of series is in H.P. Let's take a series 1/5, 1/10, 1/15, 1/20, 1/25 and check whether it is a harmonic progression or not. Below is the implementation:

```
// CPP program to check if a given
```

```
// array can form harmonic progression
#include<bits/stdc++.h>
using namespace std;
bool checkIsHP(vector<double> &arr)
{
   int n = arr.size();

   if (n == 1)
   {
      return true;
   }

   // Find reciprocal of arr[]
   vector<int> rec;
   for (int i = 0; i < n; i++)
   {
      rec.push_back((1 / arr[i]));
   }

   // return (rec);

   // After finding reciprocal, check if
   // the reciprocal is in A. P.
   // To check for A.P., first Sort the
   // reciprocal array, then check difference
   // between consecutive elements
   sort(rec.begin(), rec.end());
   int d = (rec[1]) - (rec[0]);
   for (int i = 2; i < n; i++)
   {
      if (rec[i] - rec[i - 1] != d)
      {
         return false;
      }
   }
   return true;
}
```

```
// Driver Code
int main()
{
    // series to check whether it is in H.P
    vector<double> arr = {1 / 5, 1 / 10, 1 / 15, 1 / 20, 1 / 25};

    // Checking a series is in H.P or not
    if (checkIsHP(arr))
    {
        cout << "Yes" << std::endl;
    }
    else
    {
        cout << "No" <<endl;
    }
    return 0;
}
// This code is contributed by mits
```

Output:

Yes

CONCLUSION

After comparing these methods we can conclude that they are all highly accurate with the spectral method outperforming the CFD in terms of computational speed. The SQLM though was less accurate for the three-dimensional problems. The CFD-QLM was able to handle the three-dimensional problem. We also observed that the CFD-QLM converges faster than the CFD-RM. When solving nonlinear boundary value problems, we recommend that the spectral-method-based algorithms must be used in problems involving small systems of equations and when computational speed is of importance because they are easy to code and computationally faster. CFD-based algorithms are ideal for solving large systems of equations when high accuracy is required.

The analysis of SEM shows that the first-order SEM has the same dispersion as DFDM and it is therefore not suitable for wave propagation in models with fluid-solid interfaces. This phenomenon is alleviated by using higher-order basis functions, which in fact reduces the amount of dispersion error and numerical anisotropy to a minimum.

Variational formulations based on action-type functionals differ substantially from formulations encountered in thermodynamics of Onsager

and Prigogine. The method of variational potentials (applicable to various L) may provide a relation between these two types of variational settings. The theory of a limiting reversible process may serve as a basis and indicator when choosing a suitable kinetic potential. The changes caused by the irreversibility imply the necessity to adjoint to the kinetic potential both sort of equations: those describing irreversible kinetics and those representing balance or conservation laws.

This chapter has given a comprehensive overview of convergence, convergence rates, and ways to accurately judge these aspects in a finite element analysis. Convergence plays an important role in the accuracy of solutions obtained using numerical techniques like FEA, and thus needs to be comprehensively analyzed in any given problem.

REFERENCES

- Babuška, Ivo; Banerjee, Uday; Osborn, John E. (June 2004). Generalized Finite Element Methods: Main Ideas, Results, and Perspective. International Journal of Computational Methods. 1 (1): 67–103. doi:10.1142/S0219876204000083.
- Bathe K. J. Numerical methods in finite element analysis, Prentice-Hall (1976).
- Bathe, K.J. (2006). Finite Element Procedures. Cambridge, MA: Klaus-Jürgen Bathe. ISBN 978-0979004902.
- Beidokhti Naghibi, Hamid; Janssen, Dennis; Khoshgoftar, Mehdi; Sprengers, Andre; Perdahcioglu, Emin Semih; Boogaard, Ton Van den; Verdonschot, Nico (2016). A comparison between dynamic implicit and explicit finite element simulations of the native knee joint (PDF). Medical Engineering & Physics. 38 (10): 1123–1130. doi:10.1016/j.medengphy.2016.06.001. PMID 27349493.
- Bo Strand (1994). Summation by Parts for Finite Difference Approximations for d/dx. Journal of Computational Physics. doi:10.1006/jcph.1994.1005.
- Chaskalovic J. Finite Elements Methods for Engineering Sciences, Springer Verlag, (2008).
- Christian Grossmann; Hans-G. Roos; Martin Stynes (2007). Numerical Treatment of Partial Differential Equations. Springer Science & Business Media. Courant, R. (1943). Variational methods for the solution of problems of equilibrium and vibrations. Bulletin of the American Mathematical Society. 49: 1–23. doi:10.1090/s0002-9904-1943-07818-4.
- Crank, J. The Mathematics of Diffusion. 2nd Edition, Oxford, 1975, p. 143.
- Daryl L. Logan (2011). A first course in the finite element method. Cengage Learning. ISBN 978-0495668251.
- Endre Süli: Finite Element Methods for Partial Differential Equations.
- Finite Element Analysis: How to create a great model. Coventive Composites. 2019-03-18. Retrieved 2019-04-05.
- Hastings, J. K., Juds, M. A., Brauer, J. R., Accuracy and Economy of Finite Element Magnetic Analysis, 33rd Annual National Relay Conference, April 1985.

- Hinton, Ernest; Irons, Bruce (July 1968). Least squares smoothing of experimental data using finite elements. Strain. 4 (3): 24–27. doi:10.1111/j.1475-1305.1968.tb01368.x.
- Hoffman JD; Frankel S (2001). Numerical methods for engineers and scientists. CRC Press, Boca Raton.
- Hrennikoff, Alexander (1941). Solution of problems of elasticity by the framework method. Journal of Applied Mechanics. 8 (4): 169–175.
- Jaluria Y; Atluri S (1994). Computational heat transfer. Computational Mechanics. 14: 385–386. doi:10.1007/BF00377593.
- Kiritsis, D.; Eemmanouilidis, Ch.; Koronios, A.; Mathew, J. (2009). Engineering Asset Management. Proceedings of the 4th World Congress on Engineering Asset Management (WCEAM): 591–592.
- Long Peng, Wang Jinliang; Zhu Qiding (19 May 1995). Methods with high accuracy for finite element probability computing. Journal of Computational and Applied Mathematics. 59 (2): 181–189. doi:10.1016/0377-0427(94)00027-X.
- Majumdar P (2005). Computational methods for heat and mass transfer (1st ed.). Taylor and Francis, New York.
- Mark H. Carpenter; David I. Gottlieb; Saul S. Abarbanel (1994). Time-stable boundary conditions for finite-difference schemes solving hyperbolic systems: Methodology and application to high-order compact schemes. Journal of Computational Physics. doi:10.1006/jcph.1994.1057.
- McLaren-Mercedes (2006). McLaren Mercedes: Feature - Stress to impress. Archived from the original on 2006-10-30. Retrieved 2006-10-03.
- Olek C Zienkiewicz; Robert L Taylor; J.Z. Zhu (31 August 2013). The Finite Element Method: Its Basis and Fundamentals. Butterworth-Heinemann. ISBN 978-0-08-095135-5.
- Paulsen Gard; Håkon With Andersen; John Petter Collett; Iver Tangen Stensrud (2014). Building Trust, The history of DNV 1864-2014. Lysaker, Norway: Dinamo Forlag A/S. pp. 121, 436. ISBN 978-82-8071-256-1.
- Reddy, J. N. (2006). An Introduction to the Finite Element Method (Third ed.). McGraw-Hill. ISBN 9780071267618.
- Smith GD (1985). Numerical solution of partial differential equations: finite difference methods (3rd ed.). Oxford University Press.
- Solin P., K. Segeth, I. Dolezel: Higher-Order Finite Element Methods, Chapman & Hall/CRC Press, 2003
- Song, Chongmin; Wolf, John P. (5 August 1997). The scaled boundary finite-element method – alias consistent infinitesimal finite-element cell method – for elastodynamics. Computer Methods in Applied Mechanics and Engineering. 147 (3–4): 329–355.
- Spectral Element Methods. State Key Laboratory of Scientific and Engineering Computing. Retrieved 2017-07-28.

- Strang, Gilbert; Fix, George (1973). An Analysis of The Finite Element Method. Prentice Hall. ISBN 978-0-13-032946-2.
- Thomas J.R. Hughes: The Finite Element Method: Linear Static and Dynamic Finite Element Analysis, Prentice-Hall (1987).
- What's The Difference Between FEM, FDM, and FVM?. Machine Design. 2016-04-18. Retrieved 2017-07-28.
- Zienkiewicz O. C., R. L. Taylor, J. Z. Zhu : The Finite Element Method: Its Basis and Fundamentals, Butterworth-Heinemann (2005).

MCQs

1. **The finite element method formulation of the problem results in a system of**
   a. algebraic equations
   b. logical equations
   c. Arthimatic equations
   d. flow equations
2. **FEM gives accurate representation of**
   a. real geometry
   b. complex geometry
   c. real and complex geometry
   d. constant geometry
3. **Finite element method is also called**
   a. infinite element analysis
   b. frequency element analysis
   c. finite element analysis
   d. partial element analysis
4. **Numerical algorithms are based on**
   a. FEM and FDTD
   b. FEM and IFEM
   c. TD and FD
   d. FEM and FD
5. **To solve the FEM problem, it subdivides a large problem into smaller, simpler parts that are called**
   a. finite elements
   b. infinite elements

c. dynamic elements

d. static elements

6. **Dimension of global stiffness matrix is _____**

a. N X N, where N is no of nodes

b. M X N, where M is no of rows and N is no of columns

c. Linear

d. Eliminated

7. **Each node has only _____**

a. Two degrees of freedom

b. One degree of freedom

c. Six degrees of freedom

d. Three degrees of freedom

8. **Global stiffness K is a_____ matrix.**

a. Identity matrix

b. Upper triangular matrix

c. Lower triangular matrix

d. Banded matrix

9. **In a static structural type Boundary Value Problem, at any fixed support, How many non-zero Degrees Of Freedom exist?**

a. 0

b. 1

c. 2

d. 3

10. **The dimension of Kbanded is _____ (Here NBW is half bandwidth)**

a. [N X NBW]

b. [NBW X N]

c. [N X N]

d. [NBW X NBW]

ANSWERS

1 (a)	2 (b)	3 (c)	4 (a)	5 (a)	6 (a)
7 (b)	8 (d)	9 (a)	10 (a)		

4 Finite Element Meshes

OBJECTIVES

After studying this chapter, you will be able to know:

- Choice of Mesh
- Mesh Data in Numerical Form
- Generation of Mesh Data
- Mesh Modification

1. INTRODUCTION

Engineers and scientists use finite element analysis (FEA) software to build predictive computational models of real-world scenarios. The use of FEA software begins with a computer-aided design (CAD) model that represents the physical parts being simulated as well as knowledge of the material properties and the applied loads and constraints. This information enables the prediction of real-world behavior, often with very high levels of accuracy.

The accuracy that can be obtained from any FEA model is directly related to the finite element mesh that is used. The finite element mesh is used to subdivide the CAD model into smaller domains called elements, over which a set of equations are solved. These equations approximately represent the governing equation of interest via a set of polynomial functions defined over each element. As these elements are made smaller and smaller, as the mesh is refined, the computed solution will approach the true solution.

This process of mesh refinement is a key step in validating any finite element model and gaining confidence in the software, the model, and the results. Meshes are used for rendering to a computer screen and for physical simulation such as finite element analysis or computational fluid dynamics. Meshes are composed of simple cells like triangles because, e.g., we know how to perform operations such as finite element calculations (engineering) or ray tracing (computer graphics) on triangles, but we do not know how to perform these operations directly on complicated spaces and shapes such as a roadway bridge. We can simulate the strength of the bridge, or draw it on a computer screen, by performing calculations on each triangle and calculating the interactions between triangles.

A major distinction is between structured and unstructured meshing. In structured meshing the mesh is a regular lattice, such as an array, with

implied connectivity between elements. In unstructured meshing, elements may be connected to each other in irregular patterns, and more complicated domains can be captured. This page is primarily about unstructured meshes. While a mesh may be a triangulation, the process of meshing is distinguished from point set triangulation in that meshing includes the freedom to add vertices not present in the input. "Facetting" (triangulating) CAD models for drafting has the same freedom to add vertices, but the goal is to represent the shape accurately using as few triangles as possible and the shape of individual triangles is not important. Computer graphics renderings of textures and realistic lighting conditions use meshes instead.

Many mesh generation software is coupled to a CAD system defining its input, and simulation software for taking its output. The input can vary greatly but common forms are Solid modeling, Geometric modeling, NURBS, B-rep, STL or a point cloud.

2. CHOICE OF MESH

The mesh-connected architecture is widely used in multi-computers and computer networks for parallel and distributed computing. In such architectures each executor (processor or computer) has up to four neighbors. This condition can be naturally (physically) met in dedicated supercomputers, but the problem arises when there is a need to implement this architecture for computer networks, such as Internet, where each computer is connected to each other.

The aim of this work is to provide an algorithm to create an efficient mesh-structure, choosing the best possible connections between considered nodes of a computer network. The proposed algorithms AGAMS is based on the genetic algorithm idea. Some results of investigations of properties of AGAMS are discussed.

3. MESH DATA IN NUMERICAL FORM

A service mesh is a software infrastructure that facilitates communications between microservices. It is used to control how different parts of an application integrate with each other. This prevents monolithic application architecture by allowing applications to be broken down to a collection of services that can be built, maintained and improved independent of the others. Service meshes have changed the way that we deliver software. However, when we look at the evolution in data access, it has not undergone the same level of evolution.

Today's applications are moving to the cloud. However, these cloud applications often rely on data that is not moving to the cloud. The data set may be too large...it may be changing frequently on-premise...it may be streaming in real-time from devices...or it may be owned and controlled by

another organization. Regardless of the reason, when this data cannot migrate with the application that depends on it, connectivity challenges occur.

Whether building virtual data catalogs for analytics, AI applications or serving up data for a multi-tenant cloud application, the challenge remains the same....software providers must securely connect to this data no matter where it resides.

Functioning similar to a service mesh, a data mesh creates a layer of connectivity that abstracts away the complexities of connecting, managing and supporting access to data. At its core, It is used to stitch together data held across multiple data silos. The premise of a data mesh is that it is used to connect distributed data across different locations and organizations.

A data mesh ensures that data is highly available, easily discoverable, secure, and interoperable with the applications that need access to it.

Data meshes are used in a variety of circumstances:

- Connecting cloud applications to sensitive data that lives in a customer's on-premise or cloud environment
- Creating virtual data catalogs from a variety of data sources that can't be centralized
- Creating virtual data warehouses or data lakes for analytics and machine learning training without consolidating data into a single repository
- Giving application developers and DevOps teams ways to query data from a variety of data stores without having to think about 'how' they are accessing that data

Core Principles and Logical Architecture of Data Mesh

Data mesh objective is to create a foundation for getting value from analytical data and historical facts at scale - scale being applied to constant change of data landscape, proliferation of both sources of data and consumers, diversity of transformation and processing that use cases require, speed of response to change. To achieve this objective, I suggest that there are four underpinning principles that any data mesh implementation embodies to achieve the promise of scale, while delivering quality and integrity guarantees needed to make data usable : (1) domain-oriented decentralized data ownership and architecture, (2) data as a product, (3) self-serve data infrastructure as a platform, and (4) federated computational governance.

While I expect the practices, technologies and implementations of these principles vary and mature over time, these principles remain unchanged.

I have intended for the four principles to be collectively necessary and sufficient; to enable scale with resiliency while addressing concerns

around siloeing of incompatible data or increased cost of operation. Let's dive into each principle and then design the conceptual architecture that supports it.

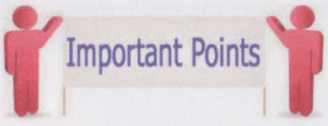
Data as a Product

One of the challenges of existing analytical data architectures is the high friction and cost of discovering, understanding, trusting, and ultimately using quality data. If not addressed, this problem only exacerbates with data mesh, as the number of places and teams who provide data - domains - increases. This would be the consequence of our first principle of decentralization. Data as a product principle is designed to address the data quality and age-old data silos problem; or as Gartner calls it dark data - "the information assets organizations collect, process and store during regular business activities, but generally fail to use for other purposes". Analytical data provided by the domains must be treated as a product, and the consumers of that data should be treated as customers - happy and delighted customers.

These measures include data quality, decreased lead time of data consumption, and in general data user satisfaction through net promoter score. Domain data product owner must have a deep understanding of who the data users are, how do they use the data,and what are the native methods that they are comfortable with consuming the data. Such intimate knowledge of data users results in design of data product interfaces that meet their needs. In reality, for majority of data products on the mesh, there a few conventional personas with their unique tooling and expectations, data analysts and data scientists. All data products can develop standardized interfaces to support them. The conversation between users of the data and product owners is a necessary piece for establishing the interfaces of data products.

Each domain will include data product developer roles, responsible for building, maintaining and serving the domain's data products. Data product developers will be working alongside other developers in the domain. Each domain team may serve one or multiple data products. It's also possible to form new teams to serve data products that don't naturally fit into an existing operational domain.

Note: this is an inverted model of responsibility compared to past paradigms. The accountability of data quality shifts upstream as close to the source of the data as possible.



Service meshes have changed the way that we deliver software.

Logical Architecture: Data Product the Architectural Quantum

Architecturally, to support data as a product that domains can autonomously serve or consume, data mesh introduces the concept of data product as its architectural quantum. Architectural quantum, as defined by Evolutionary Architecture, is the smallest unit of architecture that can be independently deployed with high functional cohesion, and includes all the structural elements required for its function.

Data product is the node on the mesh that encapsulates three structural components required for its function, providing access to domain's analytical data as a product.

- **Code:** it includes (a) code for data pipelines responsible for consuming, transforming and serving upstream data - data received from domain's operational system or an upstream data product; (b) code for APIs that provide access to data, semantic and syntax schema, observability metrics and other metadata; (c) code for enforcing traits such as access control policies, compliance, provenance, etc.
- **Data and Metadata**: well that's what we are all here for, the underlying analytical and historical data in a polyglot form. Depending on the nature of the domain data and its consumption models, data can be served as events, batch files, relational tables, graphs, etc., while maintaining the same semantic. For data to be usable there is an associated set of metadata including data computational documentation, semantic and syntax declaration, quality metrics, etc; metadata that is intrinsic to the data e.g. its semantic definition, and metadeta that communicates the traits used by computational governance to implement the expected behavior e.g. access control policies.
- **Infrastructure:** The infrastructure component enables building, deploying and running the data product's code, as well as storage and access to big data and metadata.

4. GENERATION OF MESH DATA

Techniques

Many meshing techniques are built on the principles of the Delaunay triangulation, together with rules for adding vertices, such as Ruppert's



In reality, for majority of data products on the mesh, there a few conventional personas with their unique tooling and expectations, data analysts and data scientists.

algorithm. A distinguishing feature is that an initial coarse mesh of the entire space is formed, then vertices and triangles are added. In contrast, advancing front algorithms start from the domain boundary, and add elements incrementally filling up the interior. Hybrid techniques do both. A special class of advancing front techniques creates thin boundary layers of elements for fluid flow. In structured mesh generation the entire mesh is a lattice graph, such as a regular grid of squares. Structured mesh generation for regular grids is an entire field itself, with mathematical techniques applied to ensure high-polynomial-order grid lines follow the solution space smoothly and accurately. In block-structured meshing, the domain is divided into large subregions, each of which is a structured mesh. Some direct methods start with a block-structured mesh and then move the mesh to conform to the input; see Automatic Hex-Mesh Generation based on polycube. Another direct method is to cut the structured cells by the domain boundary; see sculpt based on Marching cubes.

Some types of meshes are much more difficult to create than others. Simplicial meshes tend to be easier than cubical meshes. An important category is generating a hex mesh conforming to a fixed quad surface mesh; a research subarea is studying the existence and generation of meshes of specific small configurations, such as the tetragonal trapezohedron. Because of the difficulty of this problem, the existence of combinatorial hex meshes has been studied apart from the problem of generating good geometric realizations. While known algorithms generate simplicial meshes with guaranteed minimum quality, such guarantees are rare for cubical meshes, and many popular implementations generate inverted (inside-out) hexes from some inputs.

Meshes are often created in serial on workstations, even when subsequent calculations over the mesh will be done in parallel on super-computers. This is both because of the limitation that most mesh generators are interactive, and because mesh generation runtime is typically insignificant compared to solver time. However, if the mesh is too large to fit in the memory of a single serial machine, or the mesh must be changed (adapted) during the simulation, meshing is done in parallel.

Types of Meshes

A mesh is a representation of a larger geometric domain by smaller discrete cells. Meshes are commonly used to compute solutions of partial differential equations and render computer graphics, and to analyze geographical and cartographic data. A mesh partitions space into elements (or cells or zones) over which the equations can be solved, which then approximates the solution over the larger domain. Element boundaries may be constrained to lie on internal or

external boundaries within a model. Higher-quality (better-shaped) elements have better numerical properties, where what constitutes a "better" element depends on the general governing equations and the particular solution to the model instance.

Common Cell Shapes

Two-dimensional

There are two types of two-dimensional cell shapes that are commonly used. These are the triangle and the quadrilateral.

Computationally poor elements will have sharp internal angles or short edges or both.

Triangle

This cell shape consists of 3 sides and is one of the simplest types of mesh. A triangular surface mesh is always quick and easy to create. It is most common in unstructured grids.

Quadrilateral

This cell shape is a basic 4 sided one as shown in the figure. It is most common in structured grids.

Quadrilateral elements are usually excluded from being or becoming concave.

Three-dimensional

The basic 3-dimensional element are the tetrahedron, quadrilateral pyramid, triangular prism, and hexahedron. They all have triangular and quadrilateral faces. Extruded 2-dimensional models may be represented entirely by the prisms and hexahedra as extruded triangles and quadrilaterals. In general, quadrilateral faces in 3-dimensions may not be perfectly planar. A nonplanar quadrilateral face can be considered a thin tetrahedral volume that is shared by two neighboring elements.

Tetrahedron

A tetrahedron has 4 vertices, 6 edges, and is bounded by 4 triangular faces. In most cases a tetrahedral volume mesh can be generated automatically.

IMPORTANT

A special class of advancing front techniques creates thin boundary layers of elements for fluid flow.

Pyramid

A quadrilaterally-based pyramid has 5 vertices, 8 edges, bounded by 4 triangular and 1 quadrilateral face. These are effectively used as transition elements between square and triangular faced elements and other in hybrid meshes and grids.

Triangular prism

A triangular prism has 6 vertices, 9 edges, bounded by 2 triangular and 3 quadrilateral faces. The advantage with this type of layer is that it resolves boundary layer efficiently.

Hexahedron

A hexahedron, a topological cube, has 8 vertices, 12 edges, bounded by 6 quadrilateral faces. It is also called a hex or a brick. For the same cell amount, the accuracy of solutions in hexahedral meshes is the highest.

The pyramid and triangular prism zones can be considered computationally as degenerate hexahedrons, where some edges have been reduced to zero. Other degenerate forms of a hexahedron may also be represented.

Advanced Cells (Polyhedron)

A polyhedron (dual) element has any number of vertices, edges and faces. It usually requires more computing operations per cell due to the number of neighbours (typically 10).[2] Though this is made up for in the accuracy of the calculation.

Mesh Quality

A mesh is considered to have higher quality if a more accurate solution is calculated more quickly. Accuracy and speed are in tension. Decreasing the mesh size always increases the accuracy but also increases computational cost.

Accuracy depends on both discretization error and solution error. For discretization error, a given mesh is a discrete approximation of the space, and so can only provide an approximate solution, even when equations are solved exactly. (In computer graphics ray tracing, the number of rays fired is another source of discretization error.) For solution error, for PDEs many iterations over the entire mesh are required. The calculation is terminated early, before the equations are solved exactly. The choice of mesh element type affects both discretization and solution error.

Accuracy depends on both the total number of elements, and the shape of individual elements. The speed of each iteration grows (linearly) with the number of elements, and the number of iterations needed depends on the local solution value and gradient compared to the shape and size of local elements.

Solution Precision

A coarse mesh may provide an accurate solution if the solution is a constant, so the precision depends on the particular problem instance. One can selectively refine the mesh in areas where the solution gradients are high, thus increasing fidelity there. Accuracy, including interpolated values within an element, depends on the element type and shape.

Rate of Convergence

Each iteration reduces the error between the calculated and true solution. A faster rate of convergence means smaller error with fewer iterations.

A mesh of inferior quality may leave out important features such as the boundary layer for fluid flow. The discretization error will be large and the rate of convergence will be impaired; the solution may not converge at all.

Grid Independence

A solution is considered grid-independent if the discretization and solution error are small enough given sufficient iterations. This is essential to know for comparative results. A mesh convergence study consists of refining elements and comparing the refined solutions to the coarse solutions. If further refinement (or other changes) does not significantly change the solution, the mesh is an "Independent Grid."

Deciding the Type of Mesh

If the accuracy is of the highest concern then hexahedral mesh is the most preferable one. The density of the mesh is required to be sufficiently high in order to capture all the flow features but on the same note, it should not be so high that it captures unnecessary details of the flow, thus burdening the CPU and wasting more time. Whenever a wall is present, the mesh adjacent to the wall is fine enough to resolve the boundary layer flow and generally quad, hex and prism cells are preferred over triangles, tetrahedrons and pyramids. Quad and Hex cells can be stretched where the flow is fully developed and one-dimensional.

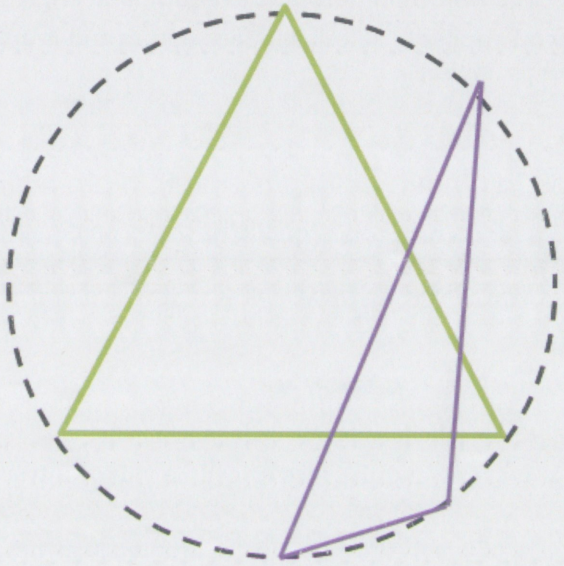


Fig. 4.1 Skewness based on equilateral volume

Based on the skewness, smoothness, and aspect ratio, the suitability of the mesh can be decided.

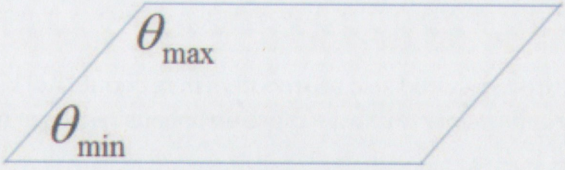


Fig. 4.2 Depicts the skewness of a quadrilateral

Skewness

The skewness of a grid is an apt indicator of the mesh quality and suitability. Large skewness compromises the accuracy of the interpolated regions. There are three methods of determining the skewness of a grid.

Based on Equilateral Volume

This method is applicable to triangles and tetrahedral only and is the default method.

$$\text{Skewness} = \frac{\text{optimal cell size - cell size}}{\text{optimal cell size}}$$

Based on the deviation from normalized equilateral angle

This method applies to all cell and face shapes and is almost always used for prisms and pyramids

$$\text{Skewness (for a quad)} = \max\left[\frac{\theta_{max} - 90}{90}, \frac{90 - \theta_{min}}{90}\right]$$

Equiangular Skew

Another common measure of quality is based on equiangular skew.

$$\text{Equiangle Skew} = \max\left[\frac{\theta_{max} - \theta_e}{180 - \theta_e}, \frac{\theta_e - \theta_{min}}{\theta_e}\right]$$

where:

θ_{max} is the largest angle in a face or cell,

θ_{min} is the smallest angle in a face or cell,

θ_e is the angle for equi-angular face or cell i.e. 60 for a triangle and 90 for a square.

A skewness' of 0 is the best possible one and a skewness of one is almost never preferred. For Hex and quad cells, skewness should not exceed 0.85 to obtain a fairly accurate solution.

For triangular cells, skewness should not exceed 0.85 and for quadrilateral cells, skewness should not exceed 0.9.

Smoothness

The change in size should also be smooth. There should not be sudden jumps in the size of the cell because this may cause erroneous results at nearby nodes.

Aspect Ratio

It is the ratio of longest to the shortest side in a cell. Ideally it should be equal to 1 to ensure best results. For multidimensional flow, it should be near to one. Also local variations in cell size should be minimal, i.e. adjacent cell sizes should not vary by more than 20%. Having a large aspect ratio can result in an interpolation error of unacceptable magnitude.

Cell Topology

Usually the cells are polygonal or polyhedral and form a mesh that partitions the domain. Important classes of two-dimensional elements include triangles (simplices) and quadrilaterals (topological squares). In three-dimensions the most-common cells are tetrahedra (simplices) and hexahedra (topological cubes). Simplicial meshes may be of any dimension and include triangles (2D) and tetrahedra (3D) as important instances. Cubical meshes is the pan-

dimensional category that includes quads (2D) and hexes (3D). In 3D, 4-sided pyramids and 3-sided prisms appear in conformal meshes of mixed cell type.

Cell Dimension

The mesh is embedded in a geometric space that is typically two or three dimensional, although sometimes the dimension is increased by one by adding the time-dimension. Higher dimensional meshes are used in niche contexts. One-dimensional meshes are useful as well. A significant category is surface meshes, which are 2D meshes embedded in 3D to represent a curved surface.

Duality

Dual graphs have several roles in meshing. One can make a polyhedral Voronoi diagram mesh by dualizing a Delaunay triangulation simplicial mesh. One can create a cubical mesh by generating an arrangement of surfaces and dualizing the intersection graph; see spatial twist continuum. Sometimes both the primal mesh and its dual mesh are used in the same simulation; see Hodge star operator. This arises from physics involving divergence and curl (mathematics) operators, such as flux & vorticity or electricity & magnetism, where one variable naturally lives on the primal faces and its counterpart on the dual faces.

Mesh Type by Use

Three-dimensional meshes created for finite element analysis need to consist of tetrahedra, pyramids, prisms or hexahedra. Those used for the finite volume method can consist of arbitrary polyhedra. Those used for finite difference methods consist of piecewise structured arrays of hexahedra known as multi-block structured meshes. 4-sided pyramids are useful to conformally connect hexes to tets. 3-sided prisms are used for boundary layers conforming to a tet mesh of the far-interior of the object.

Surface meshes are useful in computer graphics where the surfaces of objects reflect light (also subsurface scattering) and a full 3D mesh is not needed. Surface meshes are also used to model thin objects such as sheet metal in auto manufacturing and building exteriors in architecture. High (e.g., 17) dimensional cubical meshes are common in astrophysics and string theory.

Mathematical Definition and Variants

What is the precise definition of a mesh? There is not a universally-accepted mathematical description that applies in all contexts. However, some mathematical objects are clearly meshes: a simplicial complex is a mesh composed of simplices. Most polyhedral (e.g. cubical) meshes are conformal, meaning they have the cell structure of a CW complex,

a generalization of a simplicial complex. A mesh need not be simplicial because an arbitrary subset of nodes of a cell is not necessarily a cell: e.g., three nodes of a quad does not define a cell. However, two cells intersect at cells: e.g. a quad does not have a node in its interior. The intersection of two cells may be several cells: e.g., two quads may share two edges. An intersection being more than one cell is sometimes forbidden and rarely desired; the goal of some mesh improvement techniques (e.g. pillowing) is to remove these configurations. In some contexts, a distinction is made between a topological mesh and a geometric mesh whose embedding satisfies certain quality criteria.

Important mesh variants that are not CW complexes include non-conformal meshes where cells do not meet strictly face-to-face, but the cells nonetheless partition the domain. An example of this is an octree, where an element face may be partitioned by the faces of adjacent elements. Such meshes are useful for flux-based simulations. In overset grids, there are multiple conformal meshes that overlap geometrically and do not partition the domain; see e.g., Overflow, the Over set grid Flow solver. So-called meshless or meshfree methods often make use of some mesh-like discretization of the domain, and have basis functions with overlapping support. Sometimes a local mesh is created near each simulation degree-of-freedom point, and these meshes may overlap and be non-conformal to one another.

High-Order Elements

Many meshes use linear elements, where the mapping from the abstract to realized element is linear, and mesh edges are straight segments. Higher order polynomial mappings are common, especially quadratic. A primary goal for higher-order elements is to more accurately represent the domain boundary, although they have accuracy benefits in the interior of the mesh as well. One of the motivations for cubical meshes is that linear cubical elements have some of the same numerical advantages as quadratic simplicial elements. In the isogeometric analysis simulation technique, the mesh cells containing the domain boundary use the CAD representation directly instead of a linear or polynomial approximation.

Mesh Improvement

Improving a mesh involves changing its discrete connectivity, the continuous geometric position of its cells, or both. For discrete changes, for



A significant category is surface meshes, which are 2D meshes embedded in 3D to represent a curved surface.

simplicial elements one swaps edges and inserts/removes nodes. The same kinds of operations are done for cubical (quad/hex) meshes, although there are fewer possible operations and local changes have global consequences. E.g., for a hexahedral mesh, merging two nodes creates cells that are not hexes, but if diagonally-opposite nodes on a quadrilateral are merged and this is propagated into collapsing an entire face-connected column of hexes, then all remaining cells will still be hexes. In adaptive mesh refinement, elements are split (h-refinement) in areas where the function being calculated has a high gradient. Meshes are also coarsened, removing elements for efficiency. The multigrid method does something similar to refinement and coarsening to speed up the numerical solve, but without actually changing the mesh.

For continuous changes, nodes are moved, or the higher-dimensional faces are moved by changing the polynomial order of elements. Moving nodes to improve quality is called "smoothing" or "r-refinement" and increasing the order of elements is called "p-refinement." Nodes are also moved in simulations where the shape of objects change over time. This degrades the shape of the elements. If the object deforms enough, the entire object is remeshed and the current solution mapped from the old mesh to the new mesh.

Practitioners

The field is highly interdisciplinary, with contributions found in mathematics, computer science, and engineering. Meshing R&D is distinguished by an equal focus on discrete and continuous math and computation, as with computational geometry, but in contrast to graph theory (discrete) and numerical analysis (continuous). Mesh generation is deceptively difficult: it is easy for humans to see how to create a mesh of a given object, but difficult to program a computer to make good decisions for arbitrary input a priori. There is an infinite variety of geometry found in nature and man-made objects. Many mesh generation researchers were first users of meshes.

Mesh generation continues to receive widespread attention, support and funding because the human-time to create a mesh dwarfs the time to set up and solve the calculation once the mesh is finished. This has always been the situation since numerical simulation and computer graphics were invented, because as computer hardware and simple equation-solving software have improved, people have been drawn to larger and more complex geometric models in a drive for greater fidelity, scientific insight, and artistic expression.

The Mesh Refinement Process

A good finite element analyst starts with both an understanding of the physics of the system that is to be analyzed and a complete description of the

geometry of the system. This geometry is represented via a CAD model. A typical CAD model will accurately describe the shape and structure, but often also contain cosmetic features or manufacturing details that can prove to be extraneous for the purposes of finite element modeling. The analyst should put some engineering judgment into examining the CAD model and deciding if these features and details can be removed or simplified prior to meshing. Starting with a simple model and adding complexity is almost always easier than starting with a complex model and simplifying it.

The analyst should also know all of the physics that are relevant to the problem, the materials properties, the loads, the constraints, and any elements that can affect the results of interest. These inputs may have uncertainties in them. For instance, the material properties and loads may not always be precisely known. It is important to keep this in mind during the modeling process, as there is no benefit in trying to resolve a model to greater accuracy than the input data admits.

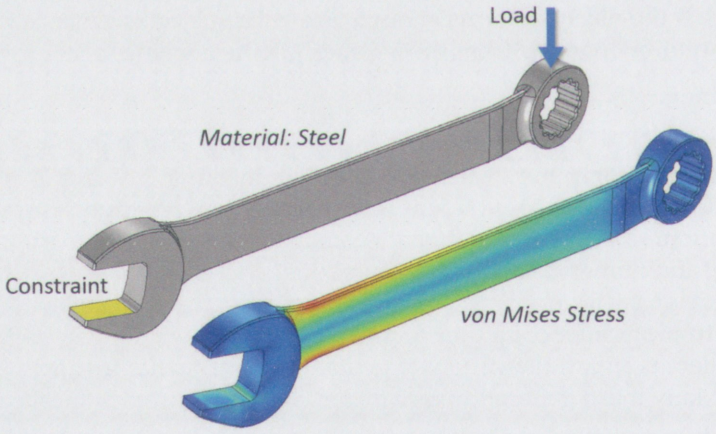


Fig. 4.3 A finite element model of a wrench and the computed stresses

Once all of this information is assembled into an FEA model, the analyst can begin with a preliminary mesh. Early in the analysis process, it makes sense to start with a mesh that is as coarse as possible – a mesh with very large elements. A coarse mesh will require less computational resources to solve and, while it may give a very inaccurate solution, it can still be used as a rough verification and as a check on the applied loads and constraints.

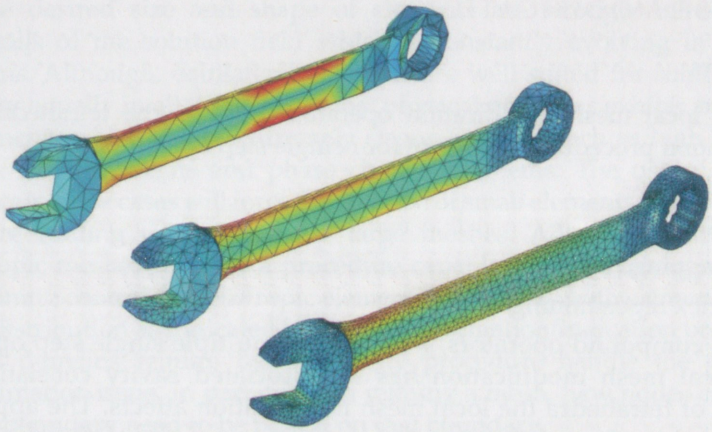


Fig. 4.4 The first few iterations of a mesh refinement study of a wrench, starting with a very coarse mesh

After computing the solution on the coarse mesh, the process of mesh refinement begins. In its simplest form, mesh refinement is the process of resolving the model with successively finer and finer meshes, comparing the results between these different meshes. This comparison can be done by analyzing the fields at one or more points in the model or by evaluating the integral of a field over some domains or boundaries.

By comparing these scalar quantities, it is possible to judge the convergence of the solution with respect to mesh refinement. After comparing a minimum of three successive solutions, an asymptotic behavior of the solution starts to emerge, and the changes in the solution between meshes become smaller. Eventually, these changes will be small enough that the analyst can consider the model to be converged. This is always a judgment call on the part of the analyst, who knows the uncertainties in the model inputs and the acceptable uncertainty in the results.

Different Mesh Refinement Metrics

Studying convergence requires choosing an appropriate mesh refinement metric. This metric can be either local or global. That is, the metric can be defined at one location in the model or as the integral of the fields over the entire model space. An example of a local metric is the displacement or stress at a point within a structural analysis. An example of a global metric is the integral of the strain energy density over all domains. Both the stresses and the strain are computed based upon the gradient of the solution and the displacement field. Gradients of the solution are always computed to one order lower polynomial approximation.

5. MESH MODIFICATION

Operators

The local mesh modification operators used in the tetrahedral mesh modification procedures consist of four single step operators:

- Splitting;
- Collapsing;
- Swapping;
- Vertex repositioning;

and compound operators, which chain multiple single step operators. Each local mesh modification has an associated cavity consisting of a number of tetrahedra the local mesh modification affects. The application of the local mesh modifications is to replace one cavity triangulation with another cavity triangulation, while the new triangulation is uniquely determined by given information or only limited configurations are possible.

Most existing procedures published in the literature present specific algorithmic application of mesh modification operators. Typical algorithms are:

- Refinement methods that control element shape by specific split orders. These procedures tend to over refine and do not consider the generalization to anisotropic case;
- Methods that coarsen only by undoing refinement. These methods cannot coarsen past the initial mesh;
- Fixed order modifications without analysis of situation. These methods do not most effectively provide the best results.

A general approach is to construct a cavity by deleting a number of tetrahedra then remesh the cavity using mesh generation algorithms (Section 5.5 discusses one such approach). Although limited use of such general approach could be acceptable to ensure solving a specific problem, any extensive use of the general approach should be avoided since it is complicated, time consuming and would make a local mesh modification procedure not as effective as would be by simply re-meshing the whole domain. Furthermore, the result mesh configuration of the general approach cannot be effectively predicted.

This section introduces the four single step operators and compound operators. It should be indicated that all single step operators are taken from previous works, so interested reader can also consult other references, for example. Compound operations are new contribution of this thesis.

The desired size and shape of elements are strongly influenced by the details of the solution field which is constantly evolving in transient problems. Although, equilateral elements are well suited for solution fields that vary equally in all directions, most physical problems exhibit anisotropic phenomena, which can be extremely strong in cases such as high Reynolds number flow problems and phase change problems. The use of isotropic elements in these cases will force the creation of small elements in all directions, therefore leading to prohibitively large meshes. Adaptively constructing anisotropic meshes requires a procedure capable of effectively creating and maintaining control of anisotropic elements that satisfy both element size and shape distribution as indicated by the error estimation/indication procedures. Curved domains require the adaptivity procedure handle the geometric approximation issue. In particular, in refining a mesh, new nodes created on curved boundary need to be placed on that boundary.

The current mesh adaptivity techniques can be decomposed into three categories:

- Adaptive re-meshing methods;
- Element subdivision methods;
- Fixed order mesh modification procedures.

Adaptive re-meshing methods construct an adaptive mesh by regenerating the entire mesh through the application of automatic mesh generation algorithms governed by desired element size and shape information. For example, Peraire et al. and Moller et al. have used adaptive re-meshing based on anisotropic advancing front mesh generation technology and the Inria group applied adaptive re-meshing in terms of anisotropic delaunay kernel. The benefit of adaptive re-meshing is that curved domains can be considered by the mesh generation algorithm and mesh coarsening is accounted for. However, it is inefficient for refining only a few elements in an already refined mesh, and introduces complexities in the transfer of solution fields from one mesh to another.

Element subdivision methods can control element shape by specific split orders, for example, the longest-edge refinement based algorithm by Rivara, the successive bisection procedure that controls split orders in terms of an affine transformation by Liu and Joe, the red–green algorithm and newest vertex bisection algorithm. Although, effective, these methods tend to over refine and do not consider curved geometry domains and the generalization of changing elements into desired shape. The coarsening process of these algorithms is limited to undoing refinements, therefore, they cannot coarsen past the initial mesh. The third mesh adaptation techniques apply local mesh modifications in a fixed order.

Researchers have improved the quality of an existing mesh using procedures consisting of four local mesh modification operations: swap, collapse, split and relocation. Researchers have made applications of these four local mesh modification operations governed by a desired mesh size and shape distribution for adaptivity purpose in two dimension and recently in three dimensions. The advantage of the third methods is that it is a local process in which curved geometry domains can be accounted for, collapse-based mesh coarsening can be applied, and the transfer of solution fields can be executed incrementally as each modification is applied. However, in three dimensions, the effectiveness of local mesh modification procedure in terms of efficiency and quality is a strong function of the manner in which various mesh modification operations are considered.

The purpose of the paper is to present a general mesh adaptation procedure that applies mesh modification operations to yield a mesh of the same quality as would be obtained by an adaptive anisotropic remeshing procedure but at less computational cost. In particular, given:

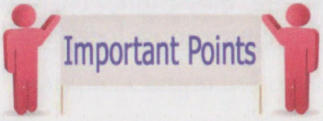
- A general 3D non-manifold domain;
- Any tetrahedral mesh of the domain;
- A desired element size and shape distribution throughout the domain;

Apply mesh modification operations effectively until the given element size and shape distribution is satisfied with the curved domain boundaries properly approximated.

CONCLUSION

Data mesh follows distributed system architecture; a collection of independent data products, with independent lifecycle, built and deployed by likely independent teams. However for the majority of use cases, to get value in forms of higher order datasets, insights or machine intelligence there is a need for these independent data products to interoperate; to be able to correlate them, create unions, find intersections, or perform other graphs or set operations on them at scale. For any of these operations to be possible, a data mesh implementation requires a governance model that embraces decentralization and domain self-sovereignty, interoperability through global standardization, a dynamic topology and most importantly automated execution of decisions by the platform.

Meshing is one of the key components to obtaining accurate results from an FEA model. The elements in the mesh must take many aspects into account to



The use of isotropic elements in these cases will force the creation of small elements in all directions, therefore leading to prohibitively large meshes.

be able to discretize stress gradients accurately. Typically, the smaller the mesh size, the more accurate the solution as the designs are better sampled across the physical domains. The trade-off is that the higher the accuracy, the larger the simulations become and thus solve times is extended. There is no point in spending extra hours running a simulation with a dense mesh if a coarser mesh will give you the results you need! Engineers often perform convergence studies to obtain the optimal balance between accuracy and solve time.

REFERENCES

- Bern, Marshall; Plassmann, Paul (2000), "Mesh Generation", Handbook of Computational Geometry. Elsevier Science.
- Castillo, J.E. (1991), "Mathematical aspects of grid Generation", Society for Industrial and applied Mathematics, Philadelphia.
- Edelsbrunner, Herbert (2001), "Geometry and Topology for Mesh Generation", Applied Mechanics Reviews, Cambridge University Press, 55 (1): B1–B2.
- Frey, Pascal Jean; George, Paul-Louis (2000), Mesh Generation: Application to Finite Elements, Hermes Science, ISBN 978-1-903398-00-5.
- George, P.L. (1991), Automatic Mesh Generation.
- Jan Brandts, Sergey Korotov, Michal Krizek: "Simplicial Partitions with Applications to the Finite Element Method", Springer Monographs in Mathematics,ISBN 978-3030556761 (2020). url="https://www.springer.com/gp/book/9783030556761".
- Mavriplis, D.J. (1996), "Mesh Generation and adaptivity for complex geometries and flows", Handbook of Computational Fluid Mechanics.
- Meshing, Lecture 7". Andre Bakker. Retrieved 2012-11-10.
- Oden, J.Tinsley; Cho, J.R. (1996), "Adaptive hpq-Finite Element Methods of Hierarchical Models for Plate- and Shell-like Structures", Computer Methods in Applied Mechanics and Engineering, 136 (3): 317–345, Bibcode:1996CMAME.136..317O, doi:10.1016/0045-7825(95)00986-8.
- Sritharan S. S. (1992), "Theory of Harmonic Grid Generation-II", Applicable Analysis, 44 (1): 127–149, doi:10.1080/00036819208840072.
- Shimada, Kenji; Gossard, David C. (1995), Bubble Mesh: Automated Triangular Meshing of Non-Manifold Geometry by Sphere Packing, ACM, pp. 409-419, doi:10.1145/218013.218095, ISBN 0-89791-672-7.
- Steven J. Owen (1998), A Survey of Unstructured Mesh Generation Technology, pp. 239–267.

- Thompson, J. F.; Warsi, Z. U. A.; Mastin, C. W. (1985), Numerical Grid Generation: Foundations and Applications, North-Holland, Elsevier.

MCQs

1. **In Finite Element Method (FEM), which option is not correct with respect to discretization or mesh generation?**
   a. Density of elements depends on the degree of accuracy desired
   b. Choice of element type may depend on the geometry of domain
   c. For better accuracy correct formulae has to be applied
   d. Some general rules govern mesh generation
2. **In Finite Element Method (FEM), Mesh convergence is a measure of the number of _____ required in a model to ensure that the results of an analysis are not affected by changing the size of the _____**
   a. elements, mesh
   b. nodes, mesh
   c. nodes, element
   d. degrees of freedom, element
3. **In Finite Element Method (FEM), While domain discretization, at which of the following geometrical cases a node is not required?**
   a. Change in cross-section area only
   b. Change in cross-section only
   c. Change in cross section area as well as change in cross-section
   d. No change
4. **In Finite Element Method (FEM), In order to increase accuracy of a solution, while domain discretization, more than one node is to be placed at a point source.**
   a. True
   b. False
5. **In Finite Element Method (FEM), for discretization purpose in general, a refined mesh is employed in areas where changes in geometry, boundary conditions, loading, material properties or solution are present.**
   a. True
   b. False

ANSWERS

1 (c) 2 (c) 3 (a) 4 (d) 5 (b)

5 Some Harmonic Problems

OBJECTIVES

After studying this chapter, you will be able to know:

- Case Study: Downstream Viscous Flow in a Rectangular Channel
- Case Study: Torsion of Prismatic Bars

1. INTRODUCTION

Domestic and industrial loads contain increasing numbers of electronic circuits that are supplied with currents that are not purely sinusoidal. For example, engines increasingly use frequency regulation, which requires converting alternating current (AC) to direct current (DC) and then DC back to AC. Given that the supply is normally in AC, this requires increasing use of electronic power converters (rectifiers, inverters, etc.) for these DC-AC and AC-DC transformations. The same thing happens with common loads like computers, LED and discharge lighting, lifts, etc.

This means that the electrical network must supply a large number of charges that rectify the current, distorting the wave shape of the current being consumed so that it is not a purely sinusoidal wave but rather a superposition of sinusoidal waves with frequencies that are multiples of the network frequency (harmonics). This type of current is most abundant in installations like offices, shopping centres, hospitals, etc., and it is formed by a 50 or 60 Hz component (network fundamental frequency) and a set of different percentages of components with multiple frequencies. These percentages may be measured using a harmonics analyzer, which can also measure the total distortion rate, or THD, which gives the ratio between the effective value of the ripple and the effective value of the fundamental component.

The result of non-sinusoidal consumption is that the voltage also suffers some distortion due to the voltage drops in the impedances of lines and transformers. In the logs we can note a slight voltage distortion in the single-phase network (low THand a stronger distortion in the three-phase example. In both cases the current shape differs greatly from the sinusoidal current, and the THD values are higher.

To regulate this issue and limit the voltage distortion levels at connection points to public networks, there are international standards that establish harmonic emission limits for units and systems connected to the network. The most significant are those associated with compatibility levels.

2. CASE STUDY: DOWNSTREAM VISCOUS FLOW IN A RECTANGULAR CHANNEL

Downstream conditions for a rectangular concrete channel (b = 10 ft) are forcing a hydraulic jump. If Q = 100 cfs and the upstream Froude number is 2.5, find the horsepower lost in the jump. Assume steady flow.

Solution:

The Froude number is defined as

$$Fr = \frac{V}{(gy)^{\frac{1}{2}}}$$

where V = Q/A = Q/yb

Rearranging the equation to yield

$$Fr = \frac{Q}{yb}\frac{1}{(gy)^{\frac{1}{2}}}$$

$$\Rightarrow y^{\frac{3}{2}} = \frac{Q}{bFr\sqrt{g}}$$

The upstream water depth and velocity are determined as follows:

$$y_1\left[\frac{Q}{bFr\sqrt{g}}\right]^{\frac{2}{3}} = \left[\frac{100}{(10)(2.5)\sqrt{32.2}}\right]^{\frac{2}{3}}$$

$$V_1 = \frac{Q}{yb} = \frac{100}{(0.792)(10)} = 12.625\,ft/S$$

Using the following equation derived from the conservation of momentum, the depth after the jump can be calculated (for rectangular channel only):

$$\frac{2y_2}{y_1} = -1\left(1+8Fr_1^{\,2}\right)^{\frac{1}{2}}$$

$$y_2 = \frac{1}{2}y_1\left[-1+\left(1+8(2.5)^2\right)^{\frac{1}{2}}\right]$$

$$= \frac{1}{2}(0.792)(6.14143)$$

$$= 2.432\,ft$$

$V_1y_1 = V_2y_2$ for steady flow, thus

$$V_2 = \frac{V_1y_1}{y_2} = \frac{(12.625)(0.792)}{(2.432)} = 4.11\,ft/s$$

The dissipation head loss across the jump is given by (assuming flat bottom channel, i.e., $\Delta z = 0$)

$$hf = E_1 - E_2 = \left(y_1 + \frac{V_1^2}{2g}\right) - \left(y_2 + \frac{V_2^2}{2g}\right)$$

Using the sequent depth relationship for rectangular channel

$$Fr_1^2 = \frac{1y_2}{2y_1}\left(\frac{y_2}{y_1} + 1\right)$$

then

$$h_f = \frac{(y_2 - y_1)^3}{4y_1y_2} = \frac{(2.432 - 0.792)^3}{4(2.432)(0.792)}$$

Power dissipated	=	ϱgQh
	=	(62.4 lb/ft3)(100 cfs)(0.57265 ft)
	=	3,573 lb-ft/s
Since 1 hp	=	550 ft-lb/s,
Power dissipated	=	3,573/550 = 6.5 hp

Separation and Wakes

An important observation is that fluid flow is always irreversible. It is irreversible because fluids have viscosity, and whenever velocity gradients appear in the flow there will be friction and energy dissipation due to viscous stresses. The flow can be reversible only if there are no velocity gradients anywhere, since that is the only condition under which there is no friction. However, when frictional effects are small, a flow may be approximately reversible.

For instance, in the flow through a large duct where the boundary layers are very thin, viscous effects are confined to a rather small region, and the fluid friction may sometimes be neglected. This is not the case for most practical flows. In most pipe and duct flows, for example, the velocity gradients extend over the entire cross-section and frictional stresses are all important. Even if the boundary layers are thin to begin with, they can thicken rapidly under some circumstances, and the flow can **separate**, as in the diffuser flow shown in figure.

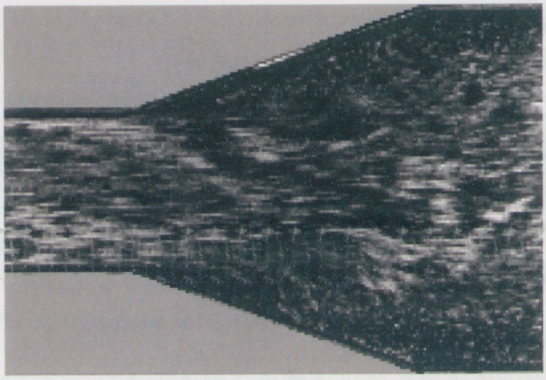


Fig. 5.1 Separated diffuser flow (included angle =20°). Flow is from left to right.

If the flow is in the direction of increasing area (this is called a diffuser flow), the effects of friction and separation are usually small, if the angle of divergence is very small. For larger angles (an included angle greater than about 7° is sufficient), there is a possibility of flow separation, by which we mean that in some parts of the flow the fluid is actually going in a direction opposite to the bulk flow direction. Large losses can occur, as the mechanical energy of the fluid is used to drive large unsteady eddying motions which eventually dissipate into heat. This type of flow can occur for quite small angles. In contrast, when the flow is in the direction of decreasing area, as in the contraction flow shown in figure, there is very little risk of producing large separated regions, even for very large values of the included angle (up to 45°), and generally losses are quite small.

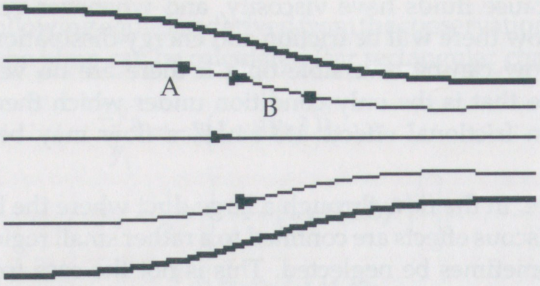
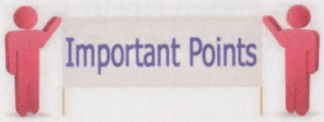


Fig. 5.2 Flow in a contraction. Flow is from left to right.



The flow can be reversible only if there are no velocity gradients anywhere, since that is the only condition under which there is no friction.

So great care needs to be taken in considering frictional effects: sometimes they are negligible, and sometimes they are not. When separation occurs, however, frictional effects are always important. It is worth noting that sharp corners almost always produce a separated flow. The sudden expansion and contraction flows shown in figure graphically demonstrate the problem, and these flows are clearly irreversible.

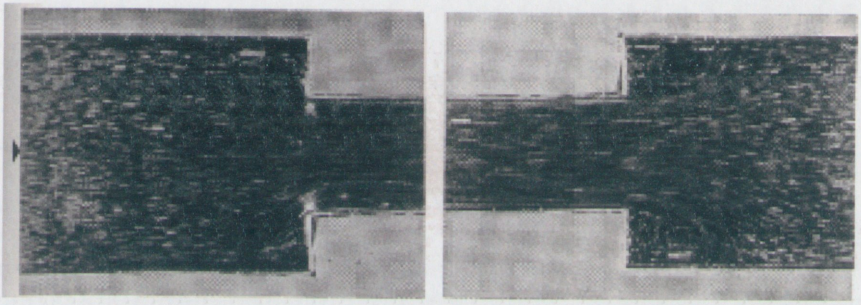


Fig. 5.3 Flow through a sudden contraction (left), and a sudden expansion (right).

Sharp corners typically produce a separated flow, but they are not the only cause of separation. For example, the flow over a cylinder can produce a large region of separated flow downstream of the cylinder.

The flow patterns over the front and back of the cylinder are quite different. In the front, the flow smoothly passes over the cylinder, but in the wake the flow is usually highly unsteady and large eddies or vortices are shed downstream. The large eddies are formed at a regular frequency and they produce pressure disturbances in the flow which we can sometimes hear as sound waves. When we talk of the wind whistling in the trees, it is the sound of eddies being shed. A Greek instrument, the Aolian harp, made use of this regular vortex shedding to produce music.

The alternate shedding of vortices produces an alternating lift force, oscillating in direction and magnitude. If the frequency of the shedding couples to a natural frequency of the cylinder and its supports, large cross-stream oscillations can occur in the cylinder position. This kind of aerodynamic instability was responsible for the destruction of the Tacoma Narrows bridge, and it has led to a number of spectacular cooling tower failures in England.

Wakes almost always contain large eddying motions which are shed downstream, although they may not shed at a regular frequency. For instance, the wake of a boat often contains large eddying motions, and this region is often called a ``dead-water'' region. Similar flow patterns are observed downstream of pylons supporting a bridge.

In any case, the eddies dissipate energy friction is the source of the irreversibility of fluid flow: either through the formation of boundary layers over the surface, or large eddies in the wake (in thermodynamics we say that the entropy of the flow increases). In an imaginary fluid that has no friction (an inviscid fluid), there would be no drag, the entropy does not change, and the flow would be reversible. For example, a cylinder in the flow of an inviscid fluid has no vortex shedding and it has no drag, contrary to our practical experience.

Hydraulic Jumps in Rectangular Channels

Hydraulic jump in a rectangular channel, also known as classical jump, is a natural phenomenon that occurs whenever flow changes from supercritical to subcritical flow. In this transition, the water surface rises abruptly, surface rollers are formed, intense mixing occurs, air is entrained, and often a large amount of energy is dissipated. In other words, a hydraulic jump happens when a higher velocity, v_1, supercritical flow upstream is met by a subcritical downstream flow with a decreased velocity, v_2, and sufficient depth. Numeric models created using the standard step method or HEC-RAS are used to track supercritical and subcritical flows to determine where in a specific reach a hydraulic jump will form.

There are common hydraulic jumps that occur in everyday situations such as during the use of a household sink. There are also man-made hydraulic jumps created by devices like weirs or sluice gates. In general, a hydraulic jump may be used to dissipate energy, to mix chemicals, or to act as an aeration device.

To produce equations describing the jump, since there is an unknown energy loss, there is a need to apply conservation of momentum.To develop this equation, a general situation in which there may or may not be an energy loss between upstream and downstream, and there may or may not be some obstacle on which there is a drag force P_f is considered. however, for a simple or classic hydraulic jump the force per unit width(P_f) equals 0. From there the momentum equation, and the conjugate depths equation can be derived.

Calculations for typical parameters in simple hydraulic jumps in rectangular channels

Energy Loss

Although momentum is conserved throughout the hydraulic jump, the energy is not. There is an initial loss of energy when the flow jumps from supercritical to subcritical depths. The resulting loss of energy is equal to the

change in specific energy across the jump and is given by the equation for ΔE below. The equation below is based on the condition that y_1 and y_2 are conjugate depths.

$$\Delta E = E_1 - E_2 = \left(y_1 + \frac{q^2}{2gy_1^2}\right) - \left(y_2 + \frac{q^2}{2gy_2^2}\right) = \frac{(y_2 - y_1)^3}{4y_1y_2}$$

When looking at the critical points on the M-y diagram and what their locations tell us about the nature of the hydraulic jump, we mentioned that an increase in q would affect the energy lost in the jump. From Figure we see that increasing the flow rate decreases the difference in the upstream and downstream depth of the jump $(y_2 - y_1)$. From this we can infer that if the momentum is held to be constant, there will be a decrease in the energy lost in the jump if the flow rate is increased.

The efficiency of the jump is determined by the dimensionless parameter E_2/E_1 which tells us how much of the original energy is remaining after the jump is complete. The equation for the energy efficiency is given below and shows the heavy dependence that the efficiency has on the Froude number of the upstream flow. Example 2 shows a sample calculation for energy loss and efficiency.

$$\frac{E_2}{E_1} = \frac{(8Fr_1^2 + 1)^{3/2} - 4Fr_1^2 + 1}{8Fr_1^2(2 + Fr_1^2)}$$

Example 2: Energy loss and efficiency

Given:

Rectangular Channel

Velocity, $v = 10$ m/s

Depth, $y_1 = 0.5$ m

Find:

Energy loss and efficiency across the hydraulic jump

Solution:

$$(1)\ Fr_1 = \frac{v}{\sqrt{gy}} = \frac{10[m/s]}{\sqrt{9.81[m/s^2] * 0.5[m]}} = 4.5$$



The equation for the energy efficiency is given below and shows the heavy dependence that the efficiency has on the Froude number of the upstream flow.

(2) $Fr_1 > 1$ (supercritical flow

(3) $y_2 = \frac{y_1}{2}\left(\sqrt{1+8Fr_1^2}-1\right) = \frac{0.5}{2}\left(\sqrt{1+8*4.5^2}-1\right) = 2.94\ m$

(4) $\Delta E = \frac{(y_2-y_1)^3}{4y_1y_2} = \frac{(2.94-0.5)^3}{4*2.94*0.5} = 2.47\ m$

(5) $\frac{E_2}{E_1} = \frac{(8Fr_1^2+1)^{3/2}-4Fr_1^2+1}{8Fr_1^2(2+F_1^2)} = \frac{(8*4.5^2+1)^{3/2}-4*4.5^2+1}{8*4.5^2(2+4.5^2)} = 0.55$

(6)% efficiency = 0.55*100 = 55% efficient

Length of Hydraulic Jump

Length of a hydraulic jump is often hard to measure in the field and during laboratory investigations due to the sudden changes in surface turbulence, in addition to the formation of roller and eddies. The length of a hydraulic jump is often an important factor to know when considering the design of structures like settling basins. The equation derived for length is based on experimental data, and relates the length to the upstream Froude number.

(1) $L = 220 \times y_1 \times \tanh\frac{Fr_1-1}{22}$

Example 3: Length calculation

Given:

Use data from Example 2

Find:

Length of jump

Solution:

(2) $L = 220*0.5*\tanh\left(\frac{4.5-1}{22}\right)$

(3) $\tanh(z) = \frac{e^z-e^{-z}}{e^z+e^{-z}}$ if necessary, this is an alernate way to calculate tanh(z)

(4) $z = \frac{4.5-1}{22} = 0.1591$

(5) $\tanh(z) = 0.1578$

(6) $L = 220*0.5*0.1578 = 17.4\ m$

Height of Hydraulic Jump

The height of the hydraulic jump, similar to length, is useful to know when designing waterway structures like settling basins or spillways. The height of the hydraulic jump is simply the difference in flow depths prior to and after the hydraulic jump. The height can be determined using the Froude number and upstream energy.

Equations:

(1) $\mathbf{h_j = y_2 - y_1}$

$$(2)\ y_2 = \frac{y_1}{2}\left(\sqrt{1+8Fr_1^2} - 1\right)$$

Substitute y_2 equation into jump height equation:

$$(3)\ h_j = \frac{y_1}{2}\left(\sqrt{1+8Fr_1^2} - 1\right) - y_1$$

$$(4)\ h_j = \frac{y_1\sqrt{1+8Fr_1^2} - 3y_1}{2}$$

Example 4: Height calculation

Given:

Use data from Example 2

Find:

Height of jump

Solution:

$$(5)\ h_j = \frac{y_1\sqrt{1+8Fr_1^2} - 3y_1}{2}$$

$$(6)\ h_j = \frac{0.5\sqrt{1+8(4.5)^2} - (3*0.5)}{2}$$

(7) $\mathbf{h_j = 2.4\ m}$

3. CASE STUDY: TORSION OF PRISMATIC BARS

When a torque is applied to a beam with non-circular cross-section, the cross-section rotates about the longitudinal axis of the beam and simultaneously undergoes a significant distortion. The cross-section thus undergoes both twisting and warping deformations.

Such problems are formulated using theory of elasticity principles. The foundational assumptions of the formulation are the strain-displacement

(kinematirelations of infinitesimal/small deformation assumptions, the stress-strain laws, and the differential equations of equilibrium and compatibility requirements. Saint-Venant formulated the problem using theory of elasticity and Prandtl solved the problem in terms of Prandtl's stress functions.

Prandtl's formulation of the Saint-Venant torsion problem leads to a Poisson type partial differential equation (PDE); which can be solved using analytical or numerical methods. Available analytical techniques include the method of separation of variables, eigenfunction expansion methods, integral transform methods and Green's function methods. The numerical methods that can be used to solve the torsion problem are the numerical methods available for solving the boundary value problems (BVP) in engineering. Some of the numerical techniques for solving BVP which are applicable to the Poisson type PDE are Galerkin's variational method, Extended Galerkin's variational method, Ritz's method, Finite Element method (FEM), Finite difference method (FDM); and Boundary element methods (BEM).

In this work, the Saint-Venant problem of torsion of prismatic bars with rectangular cross-section will be formulated in variational form, and solved using the Galerkin variational method.

Research Aim and Objectives

The research aim is to use the Galerkin variational method to solve the Saint-Venant torsion problem for prismatic bars with rectangular cross-sections. The specific objectives are:

1. To formulate the Saint-Venant torsional problem for prismatic bars with rectangular cross-sections using theory of elasticity principles and techniques
2. To show that the formulated Saint-Venant torsional problem is governed by a partial differential equation called the Poisson equation when expressed in terms of the Airy's and Prandtl's stress functions of elasticity
3. To express the boundary value problem (BVP) in variational form using the Galerkin variational method
4. To solve the Galerkin variational equation for the problem and thus obtain solutions for the unknown parameters in the assumed (trial) solutions for the Airy's or Prandtl's stress function

IMPORTANT

The foundational assumptions of the formulation are the strain-displacement (kinematic) relations of infinitesimal/small deformation assumptions, the stress-strain laws, and the differential equations of equilibrium and compatibility requirements.

5. To obtain analytical expressions for the torque, shear stresses and torsional parameters
6. To show that Galerkin's variational method can be used to obtain closed form mathematical solutions to the BVP of Saint-Venant torsion for prismatic bars with rectangular cross-sections.

In this chapter, we study the stresses that develop and the displacement that these members undergo when subjected to twisting moment. While in the analysis of beams subjected to bending moments we maintained some generality with the loading, the study of torsion is focused on a single loading case - one end fixed and the other end free to twist, as shown in figure below. Here the double arrow indicates the twisting moment.

However, we study this boundary value problem for a variety of cross sections. This analysis for torsion depends on whether the section is thick walled or thin walled as in the case of bending moments. It also depends on whether the section is open or closed. We shall discuss in detail as to when a section is closed subsequently.

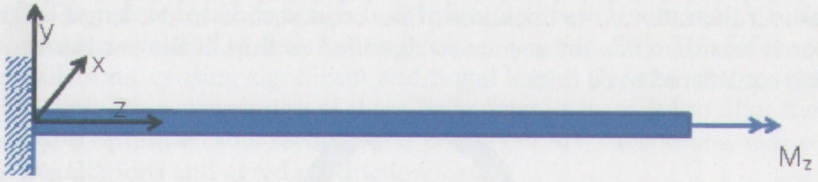


Fig. 5.4 Schematic of a straight prismatic member subjected to end torsion

Before proceeding further, we would like to point out the change in the orientation of the coordinate system, from that used in the study of the beams. This is necessitated because for some problems we would be using cylindrical polar coordinates, and want the cross section to be in the xy plane. We use cylindrical polar coordinates with this orientation so that the boundary of the cross section can be defined easily.

Since, the value of displacement or stress in a material particle is independent of the coordinate system used, it is expected that the analyst pick coordinate systems that is convenient for a problem and not that which he is comfortable with. Hence, the change.

For this orientation of the coordinate system, following the same steps as discussed in detail in chapter 8, it can be shown that the torsional moment, M_z is given by,

$$M_z = \int_a \left(\sigma_{yz} x - \sigma_{xz} y\right) dxdy,$$

and for completeness, the other two components of the moment, which are bending moments are,

$$M_y = -\int_a \sigma_{zz} x dxdy,$$

$$M_x = -\int_a \sigma_{zz} y dxdy,$$

Comparing equations (1) with (2) and (3) it can be seen that while the bending moments are due to normal stresses, torsional moment is due to shear stresses. This is the major difference between the bending and twisting moment. While the bending moment gives raise to normal stresses predominantly, twisting moment gives raise to shear stresses predominantly.

Now let us see how to classify the sections. As in case of bending, a cross section would be classified as thin walled if the thickness of the cross section is such that it is much less than the characteristic dimensions of the cross section. Typically, if the ratio of the thickness of the cross section to the length of the member is less than 0.1, the section is classified as thin. If the section is not thin, it is considered to be thick.

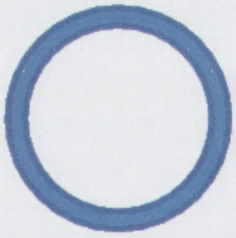


Fig. 5.5 Section classified as closed sections when subjected to torsion

If a section when twisted can deform such that plane sections before deformation remain plane after twisting is called closed section. Sections which do not deform in the above manner are called open sections. Solid Circular cross section is an example of closed section and solid rectangular cross section is an example of open sections. Similarly, a thin walled annular cylinder (see figure above) is an example of closed section and if the same annular cylinder has a longitudinal slit (see figure below), it is an example of open section. All closed section allows for continuous variation of shear stresses such that it is nowhere zero except at the centroid of the cross section.

A section which requires the shear stresses to be zero at locations apart from the centroid of the cross section, like at the corners of a rectangular cross

section, is an open section. It should be pointed out that in elliptical shaped cross section also the shear stress is zero only at the centroid of the cross section but is an open section as plane sections do not remain plane after twisting. Thus, the requirement on the shear stress is just necessary but not sufficient to classify a given section as closed.

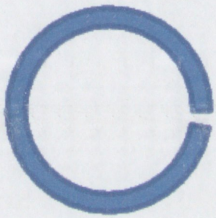


Fig. 5.6 Section classified as open sections when subjected to torsion

In the following section, we find the displacement field and stress field for twisting of a thick walled sections and then focus on thin walled sections.

CONCLUSION

The presence of harmonics in distribution lines is increasing, causing a set of problems related to deteriorating voltage wave quality. This forces oversize installations, causing significant additional losses. Despite existing standards that limit the consumption of these harmonics, it is useful to filter them as it lets you optimise cable sections and powers to MV substations, reduce losses in installations and avoid production losses.

The solution to the problem is a rational and comprehensive design of harmonic filters, like active filters, which helps solve the problem with affordable costs easily offset by savings in losses, improves the life of some of the components in installations and optimises their distribution infrastructure (cable conduits, transformers, etc.).

REFERENCES

- Akan, A. O. (2006). Open Channel Hydraulics, Elsevier Ltd., Burlington, MA.
- Chanson, Hubert (2009). "Current knowledge in hydraulic jumps and related phenomena. A survey of experimental results" (PDF). European Journal of Mechanics B. 28 (2): 191–210. Bibcode:2009EJMF...28..191C. doi:10.1016/j.euromechflu.2008.06.004.
- Chanson, Hubert (2012). "Momentum Considerations in Hydraulic Jumps and Bores" (PDF). Journal of Irrigation and Drainage Engineering. 138 (4): 382–385. doi:10.1061/(ASCE)IR.1943-4774.0000409.
- Chaudhry, M. H. (2008). Open-Channel Flow, Springer Science+Business Media, LLC, New York, NY.

- Chaudhry, M. Hanif (2007-12-04). Open-Channel Flow. ISBN 978-0-387-30174-7.
- Chow, V. T. (1959). Open-Channel Hydraulics, McGraw-Hill, New York, NY.
- Hager, W.H. 1992. Energy Dissipators and Hydraulic Jump. Kluwer Academic, Dordrecht, The Netherlands. ISBN 0-7923-1508-1.
- Khatsuria, R.M. The Hydraulics of Spillways and Energy Dissipators. ISBN 0-8247-5789-0. CRC Press, 2005].
- Surface Tension and the Hydraulic Jump". Math.mit.edu. Retrieved 2010-11-10.
- White, Frank M. (6th edition). Fluid Mechanics. McGraw Hill Inc. ISBN 0-07-293844-7.

MCQs

1. **Which of the following is used as a harmonic reduction technique in inverters?**
   a. Amplitude modulation
   b. Cycloconverter control
   c. Transformer connection
   d. Series connection of two inverters
2. **For harmonic reduction by transformer connection, the output voltages from the two inverters must be**
   a. similar and in-phase with each other
   b. dissimilar but in-phase with each other
   c. similar but phase shifted from each other
   d. dissimilar and phase shifted from each other
3. **The output voltage obtained by connecting two inverters through a transformer is a**
   a. square wave
   b. sine wave
   c. quasi-square wave
   d. none of the mentioned
4. **Pulses of different widths and heights are superimposed in case of _____ harmonic reduction technique.**
   a. transformer connection
   b. pulse width modulation
   c. stepped-wave inverter
   d. none of the mentioned
5. **In case of stepped wave inverters,**
   a. both the transformers have 1:1 turns ratio
   b. both the transformers have 1:2 turns ratio

c. both the transformers have different transformer ratio

d. none of the mentioned

6. **In stepped wave inverters, one of the inverters are gated such as to obtain**

a. one level modulation

b. two level modulation

c. zero level modulation

d. none of the mentioned

7. **In three-level modulation**

a. the output voltage is zero in the first half cycle.

b. the output voltage either zero or positive in the first half cycle.

c. the output voltage either zero or negative in the first half cycle.

d. the output voltage either zero, positive or negative in the first half cycle.

8. **In single-phase modulation of PWM inverters, the lowest harmonic can be eliminated if the pulse width is made equal to _____**

a. 30°

b. 0°

c. 120°

d. 60°

9. **In single-phase modulation of PWM inverters, the 5th order harmonic can be eliminated if the pulse width is made equal to _____**

a. 30°

b. 36°

c. 72°

d. None of the mentioned

10. **The waveform obtained by _____ is more near to a sinusoidal wave**

a. PWM inverters

b. Stepped wave inverters

c. parallel connected inverters

d. None of the mentioned

ANSWERS

1 (c)	2 (c)	3 (c)	4 (c)	5 (c)	6 (b)
7 (d)	8 (c)	9 (c)	10 (b)		

6 Finite Element Analysis of Biharmonic Problems

OBJECTIVES

After studying this chapter, you will be able to know:

- Derivation of the Element Stiffness Matrix
- Assembly of the Overall Stiffness Matrix
- Boundary Conditions
- A Computer Program for Problems of the Biharmonic Plane Strain or Plane Stress Type

1. INTRODUCTION

The analytic element method (AEM) is a numerical method used for the solution of partial differential equations. It is similar in nature to the boundary element method (BEM), as it does not rely upon discretization of volumes or areas in the modeled system; only internal and external boundaries are discretized. One of the primary distinctions between AEM and BEMs is that the boundary integrals are calculated analytically. The basic premise of the analytic element method is that, for linear differential equations, elementary solutions may be superimposed to obtain more complex solutions. A suite of 2D and 3D analytic solutions ("elements") are available for different governing equations. These elements typically correspond to a discontinuity in the dependent variable or its gradient along a geometric boundary (e.g., point, line, ellipse, circle, sphere, etc.). This discontinuity has a specific functional form (usually a polynomial in 2D) and may be manipulated to satisfy Dirichlet, Neumann, or Robin (mixed) boundary conditions. Each analytic solution is infinite in space and/or time.

Commonly each analytic solution contains degrees of freedom (coefficients) that may be calculated to meet prescribed boundary conditions along the element's border. To obtain a global solution (i.e., the correct element coefficients), a system of equations is solved such that the boundary conditions are satisfied along all of the elements (using collocation, least-squares minimization, or a similar approach). Notably, the global solution provides a spatially continuous description of the dependent variable everywhere in the infinite domain, and the governing equation is satisfied everywhere exactly except along the border of the element, where the governing equation is not strictly applicable due to the discontinuity.

The ability to superpose numerous elements in a single solution means that analytical solutions can be realized for arbitrarily complex boundary conditions. That is, models that have complex geometries, straight or curved boundaries, multiple boundaries, transient boundary conditions, multiple aquifer layers, piecewise varying properties and continuously varying properties can be solved. Elements can be implemented using far-field expansions such that model containing many thousands of elements can be solved efficiently to high precision.

The analytic element method has been applied to problems of groundwater flow governed by a variety of linear partial differential equations including the Laplace, the Poisson equation, the modified Helmholtz equation, the heat equation, and the biharmonic equations. Often theses equations are solved using complex variables which enables using mathematical techniques available in complex variable theory. A useful technique to solve complex problems is using conformal mapping which maps the boundary of a geometry, e.g. an ellipse, onto the boundary of the unit circle where the solution is known.

In the analytic element method the discharge potential and stream function, or combined the complex potential, is used. This potential links the physical properties of the groundwater system, the hydraulic head or flow boundaries, to a mathematical representation in a potential. This mathematical representation can be used to calculate the potential in terms of position and thus also solving groundwater flow problems. Elements are developed by solving the boundary conditions for either of these two properties, hydraulic head or flow boundary, which results in analytical solutions capable of dealing with numerous boundary conditions.

2. DERIVATION OF THE ELEMENT STIFFNESS MATRIX

In the finite element method for the numerical solution of elliptic partial differential equations, the stiffness matrix represents the system of linear equations that must be solved in order to ascertain an approximate solution to the differential equation.

For simplicity, we will first consider the Poisson problem

$$-\nabla^2 u = f$$

on some domain Ω, subject to the boundary condition $u = 0$ on the boundary of Ω. To discretize this equation by the finite element method, one chooses a set of basis functions $\{\varphi 1, ..., \varphi n\}$ defined on Ω which also vanish on the boundary. One then approximates

$$u \approx u^h = u_1\varphi_1 + \cdots + u_n\varphi_n.$$

The coefficients u_1, ..., u_n are determined so that the error in the approximation is orthogonal to each basis function φ_i:

$$\int_\Omega \varphi_i \cdot f\,dx = -\int_\Omega \varphi_i \nabla^2 u^h\,dx = -\sum_j \left(\int_\Omega \varphi_i \nabla^2 \varphi_j\,dx\right) u_j = \sum_j \left(\int_\Omega \nabla\varphi_i \cdot \nabla\varphi_j\,dx\right) u_j.$$

The stiffness matrix is the n-element square matrix A defined by

$$A_{ij} = \int_\Omega \nabla\varphi_i \cdot \nabla\varphi_j\,dx.$$

By defining the vector F with components $F_i = \int_\Omega \varphi_i f\,dx$, the coefficients ui are determined by the linear system Au = F. The stiffness matrix is symmetric, i.e. $A_{ij} = A_{ji}$, so all its eigenvalues are real. Moreover, it is a strictly positive-definite matrix, so that the system Au = F always has a unique solution. (For other problems, these nice properties will be lost.)

Note that the stiffness matrix will be different depending on the computational grid used for the domain and what type of finite element is used. For example, the stiffness matrix when piecewise quadratic finite elements are used will have more degrees of freedom than piecewise linear elements.

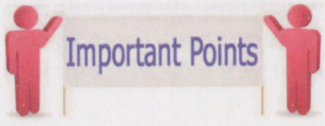
The Stiffness Matrix for Other Problems

Determining the stiffness matrix for other PDE follows essentially the same procedure, but it can be complicated by the choice of boundary conditions. As a more complex example, consider the elliptic equation

$$-\sum_{k,l} \frac{\partial}{\partial x_k}\left(a^{kl}\frac{\partial u}{\partial x_l}\right)\frac{\partial}{\partial x_l} = f$$

where $A(x) = a^{kl}(x)$ is a positive-definite matrix defined for each point x in the domain. We impose the Robin boundary condition

$$-\sum_{k,l} \nu_k a^{kl}\frac{\partial u}{\partial x_l} = c(u-g),$$



The stiffness matrix represents the system of linear equations that must be solved in order to ascertain an approximate solution to the differential equation.

where ν_k is the component of the unit outward normal vector ν in the k-th direction. The system to be solved is

$$\sum_j \left(\sum_{k,l} \int_\Omega a^{kl} \frac{\partial \varphi_i}{\partial x_k} \frac{\partial \varphi_j}{\partial x_l} dx + \int_{\partial\Omega} \alpha \varphi_i \varphi_j ds \right) u_j = \int_\Omega \varphi_i f dx + \int_{\partial\Omega} \alpha \varphi_i g ds,$$

as can be shown using an analogue of Green's identity. The coefficients u_i are still found by solving a system of linear equations, but the matrix representing the system is markedly different from that for the ordinary Poisson problem.

In general, to each scalar elliptic operator L of order 2k, there is associated a bilinear form B on the Sobolev space H^k, so that the weak formulation of the equation Lu = f is

$$B[u, v] = (f, v)$$

for all functions v in H^k. Then the stiffness matrix for this problem is

$$A_{ij} = B[\varphi_j, \varphi_i].$$

3. ASSEMBLY OF THE OVERALL STIFFNESS MATRIX

In order to implement the finite element method on a computer, one must first choose a set of basis functions and then compute the integrals defining the stiffness matrix. Usually, the domain Ω is discretized by some form of mesh generation, wherein it is divided into non-overlapping triangles or quadrilaterals, which are generally referred to as elements. The basis functions are then chosen to be polynomials of some order within each element, and continuous across element boundaries. The simplest choices are piecewise linear for triangular elements and piecewise bilinear for rectangular elements.

The element stiffness matrix $A^{[k]}$ for element T_k is the matrix

$$A_{ij}^{[k]} = \int_{T_k} \nabla \varphi_i \cdot \nabla \varphi_j dx.$$

The element stiffness matrix is zero for most values of i and j, for which the corresponding basis functions are zero within T_k. The full stiffness matrix A is the sum of the element stiffness matrices. In particular, for basis functions that are only supported locally, the stiffness matrix is sparse.

For many standard choices of basis functions, i.e. piecewise linear basis functions on triangles, there are simple formulas for the element stiffness matrices. For example, for piecewise linear elements, consider a triangle with vertices (x_1, y_1), (x_2, y_2), (x_3, y_3), and define the 2×3 matrix

$$D = \begin{bmatrix} x_3 - x_2 & x_1 - x_3 & x_2 - x_1 \\ y_3 - y_2 & y_1 - y_3 & y_2 - y_1 \end{bmatrix}.$$

Then the element stiffness matrix is

$$A^{[k]} = \frac{D^T D}{4\,\text{area}(T)}.$$

When the differential equation is more complicated, say by having an inhomogeneous diffusion coefficient, the integral defining the element stiffness matrix can be evaluated by Gaussian quadrature.

The condition number of the stiffness matrix depends strongly on the quality of the numerical grid. In particular, triangles with small angles in the finite element mesh induce large eigenvalues of the stiffness matrix, degrading the solution quality.

4. BOUNDARY CONDITIONS

The boundary element method (BEM) is a numerical computational method of solving linear partial differential equations which have been formulated as integral equations (i.e. in boundary integral form), including fluid mechanics, acoustics, electromagnetics (where the technique is known as method of moments or abbreviated as MoM), fracture mechanics, and contact mechanics.

The integral equation may be regarded as an exact solution of the governing partial differential equation. The boundary element method attempts to use the given boundary conditions to fit boundary values into the integral equation, rather than values throughout the space defined by a partial differential equation. Once this is done, in the post-processing stage, the integral equation can then be used again to calculate numerically the solution directly at any desired point in the interior of the solution domain.

BEM is applicable to problems for which Green's functions can be calculated. These usually involve fields in linear homogeneous media. This places considerable restrictions on the range and generality of problems to which boundary elements can usefully be applied. Nonlinearities can be included in the formulation, although they will generally introduce volume integrals which then require the volume to be discretised before solution can be attempted, removing one of the most often cited advantages of BEM. A useful technique for treating the volume integral without discretising the volume is the dual-reciprocity method.

The technique approximates part of the integrand using radial basis functions (local interpolating functions) and converts the volume integral into boundary integral after collocating at selected points distributed throughout the volume domain (including the boundary). In the dual-reciprocity BEM, although there is no need to discretize the volume into meshes, unknowns at

chosen points inside the solution domain are involved in the linear algebraic equations approximating the problem being considered.

The Green's function elements connecting pairs of source and field patches defined by the mesh form a matrix, which is solved numerically. Unless the Green's function is well behaved, at least for pairs of patches near each other, the Green's function must be integrated over either or both the source patch and the field patch. The form of the method in which the integrals over the source and field patches are the same is called "Galerkin's method". Galerkin's method is the obvious approach for problems which are symmetrical with respect to exchanging the source and field points. In frequency domain electromagnetics, this is assured by electromagnetic reciprocity. The cost of computation involved in naive Galerkin implementations is typically quite severe. One must loop over each pair of elements (so we get n^2 interactions) and for each pair of elements we loop through Gauss points in the elements producing a multiplicative factor proportional to the number of Gauss-points squared. Also, the function evaluations required are typically quite expensive, involving trigonometric/hyperbolic function calls. Nonetheless, the principal source of the computational cost is this double-loop over elements producing a fully populated matrix.

The Green's functions, or fundamental solutions, are often problematic to integrate as they are based on a solution of the system equations subject to a singularity load (e.g. the electrical field arising from a point charge). Integrating such singular fields is not easy. For simple element geometries (e.g. planar triangles) analytical integration can be used. For more general elements, it is possible to design purely numerical schemes that adapt to the singularity, but at great computational cost.

Of course, when source point and target element (where the integration is done) are far-apart, the local gradient surrounding the point need not be quantified exactly and it becomes possible to integrate easily due to the smooth decay of the fundamental solution. It is this feature that is typically employed in schemes designed to accelerate boundary element problem calculations.

Derivation of closed-form Green's functions is of particular interest in boundary element method, especially in electromagnetics. Specifically in the analysis of layered media, derivation of spatial-domain Green's function necessitates the inversion of analytically-derivable spectral-domain Green's function through Sommerfeld path integral. This integral can not be evaluated analytically and its numerical integration is costly due to its oscillatory and slowly-converging behaviour. For a robust analysis, spatial Green's functions are approximated as complex exponentials with methods such as Prony's method or generalized pencil of function, and the integral is evaluated with

Sommerfeld identity. This method is known as discrete complex image method.

Comparison to Other Methods

The boundary element method is often more efficient than other methods, including finite elements, in terms of computational resources for problems where there is a small surface/volume ratio. Conceptually, it works by constructing a "mesh" over the modelled surface. However, for many problems boundary element methods are significantly less efficient than volume-discretisation methods (finite element method, finite difference method, finite volume method). A good example of application of the boundary element method is efficient calculation of natural frequencies of liquid sloshing in tanks. Boundary element method is one of the most effective methods for numerical simulation of contact problems, in particular for simulation of adhesive contacts.

Boundary element formulations typically give rise to fully populated matrices. This means that the storage requirements and computational time will tend to grow according to the square of the problem size. By contrast, finite element matrices are typically banded (elements are only locally connected) and the storage requirements for the system matrices typically grow quite linearly with the problem size. Compression techniques (e.g. multipole expansions or adaptive cross approximation/hierarchical matrices) can be used to ameliorate these problems, though at the cost of added complexity and with a success-rate that depends heavily on the nature of the problem being solved and the geometry involved.

5. A COMPUTER PROGRAM FOR PROBLEMS OF THE BIHARMONIC PLANE STRAIN OR PLANE STRESS TYPE

In continuum mechanics, a material is said to be under plane stress if the stress vector is zero across a particular plane. When that situation occurs over an entire element of a structure, as is often the case for thin plates, the stress analysis is considerably simplified, as the stress state can be represented by a tensor of dimension 2 (representable as a 2×2 matrix rather than 3×3). A related notion, plane strain, is often applicable to very thick members.

Plane stress typically occurs in thin flat plates that are acted upon only by load forces that are parallel to them. In certain situations, a gently curved thin plate may also be assumed to have plane stress for the purpose of stress analysis. This is the case, for example, of a thin-walled cylinder filled with a fluid under pressure. In such cases, stress components perpendicular to the plate are negligible compared to those parallel to it.

In other situations, however, the bending stress of a thin plate cannot be neglected. One can still simplify the analysis by using a two-dimensional domain, but the plane stress tensor at each point must be complemented with bending terms.

Stress Transformation in Plane Stress and Plane Strain

Consider a point P in a continuum under a state of plane stress, or plane strain, with stress components (σ_x, σ_y, τ_{xy}) and all other stress components equal to zero (Figure below).

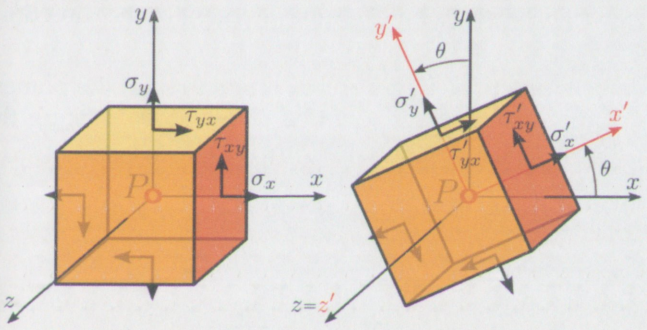


Fig. 6.1 Stress transformation at a point in a continuum under plane stress conditions.

From static equilibrium of an infinitesimal material element at P (Figure below), the normal stress σ_n and the shear stress τ_n on any plane perpendicular to the x - y plane passing through P with a unit vector n making an angle of θ with the horizontal, i.e. cos θ is the direction cosine in the x direction, is given by:

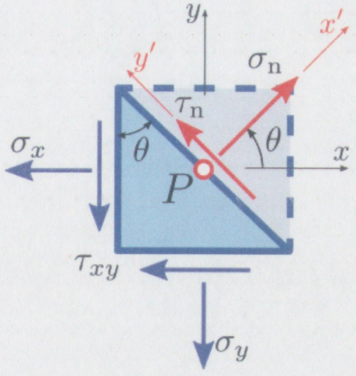


Fig. 6.2 Stress components at a plane passing through a point in a continuum under plane stress conditions.

$$\sigma_n = \frac{1}{2}(\sigma_x + \sigma_y) + \frac{1}{2}(\sigma_x - \sigma_y)\cos 2\theta + \tau_{xy} \sin 2\theta$$

$$\tau_n = -\frac{1}{2}(\sigma_x - \sigma_y)\sin 2\theta + \tau_{xy} \cos 2\theta$$

These equations indicate that in a plane stress or plane strain condition, one can determine the stress components at a point on all directions, i.e. as a function of θ, if one knows the stress components (σ_x, σ_y, τ_{xy}) on any two perpendicular directions at that point. It is important to remember that we are considering a unit area of the infinitesimal element in the direction parallel to the y - z plane.

The principal directions (Figure), i.e., orientation of the planes where the shear stress components are zero, can be obtained by making the previous equation for the shear stress τ_n equal to zero. Thus we have:

$$\tau_n = -\frac{1}{2}(\sigma_x - \sigma_y)\sin 2\theta + \tau_{xy} \cos 2\theta = 0$$

and we obtain

$$\tan 2\theta_p = \frac{2\tau_{xy}}{\sigma_x - \sigma_y}$$

This equation defines two values θ_p which are 90° apart (Figure below). The same result can be obtained by finding the angle θ which makes the normal stress σ_n a maximum, i.e. $\frac{d\sigma_n}{d\theta} = 0$

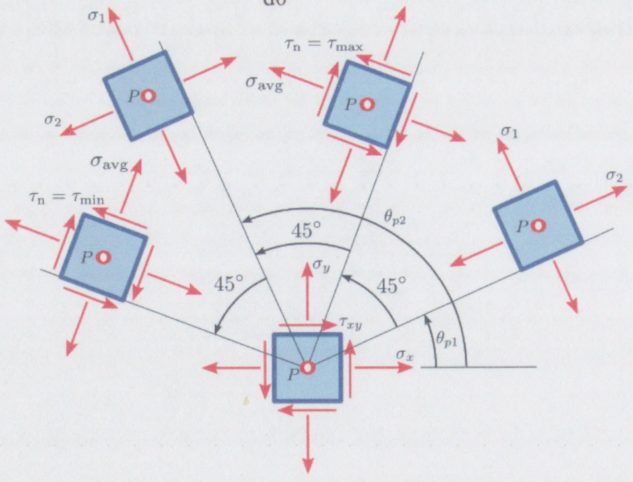


Fig. 6.3 Transformation of stresses in two dimensions, showing the planes of action of principal stresses, and maximum and minimum shear stresses

The principal stresses σ_1 and σ_2, or minimum and maximum normal stresses σ_{max} and σ_{min}, respectively, can then be obtained by replacing both values of θ_p into the previous equation for σ_n. This can be achieved by rearranging the equations for σ_n and τ_n, first transposing the first term in the first equation and squaring both sides of each of the equations then adding them. Thus we have

$$\left[\sigma_n - \tfrac{1}{2}(\sigma_x + \sigma_y)\right]^2 + \tau_n^2 = \left[\tfrac{1}{2}(\sigma_x - \sigma_y)\right]^2 + \tau_{xy}^2$$

$$(\sigma_n - \sigma_{avg})^2 + \tau_n^2 = R^2$$

where

$$R = \sqrt{\left[\tfrac{1}{2}(\sigma_x - \sigma_y)\right]^2 + \tau_{xy}^2} \quad \text{and} \quad \sigma_{avg} = \tfrac{1}{2}(\sigma_x + \sigma_y)$$

which is the equation of a circle of radius R centered at a point with coordinates [σ_{avg}, 0], called Mohr's circle. But knowing that for the principal stresses the shear stress $\tau_n = 0$, then we obtain from this equation:

$$\sigma_1 = \sigma_{max} = \tfrac{1}{2}(\sigma_x + \sigma_y) + \sqrt{\left[\tfrac{1}{2}(\sigma_x - \sigma_y)\right]^2 + \tau_{xy}^2}$$

$$\sigma_2 = \sigma_{min} = \tfrac{1}{2}(\sigma_x + \sigma_y) - \sqrt{\left[\tfrac{1}{2}(\sigma_x - \sigma_y)\right]^2 + \tau_{xy}^2}$$

CONCLUSION

The plane stress problem to be solved is a rectangular region with fixed boundaries. The finite difference model of this problem consists of two finite difference templates with even spacing. We should not expect that this simple representation will produce accurate results. This small model is designed to provide a simple introduction to the detailed formulation of a finite difference coefficient matrix.

The development of the variational form of the plane stress problem begins with the formulation of the strain energy expression. The strain energy is first formulated in terms of stresses and strains. Then, this expression is transformed to a function of strains using the constitutive relations. Finally, the strain-displacement relations are used to transform the strain energy expression to a function of u and v, with the displacements in the x and y directions, respectively.

REFERENCES

- Aksun, M. I. (February 2003). A robust approach for the derivation of closed-form Green's functions. IEEE Transactions on Microwave Theory and Techniques. 44 (5): 651–658. doi:10.1109/22.493917. hdl:11693/10779.
- Ang, Whye-Teong (2007), A Beginner's Course in Boundary Element Methods, Boca Raton, Fl: Universal Publishers, ISBN 978-1-58112-974-8.
- Ang, Whye-Teong (2013), Hypersingular Integral Equations in Fracture Analysis, Oxford: Woodhead Publishing, ISBN 978-0-85709-479-7.
- Bakker, Mark; Strack, Otto D. L. (2003-02-10). Analytic elements for multiaquifer flow. Journal of Hydrology. 271 (1): 119–129. doi:10.1016/S0022-1694(02)00319-0. ISSN 0022-1694.
- Banerjee, Prasanta Kumar (1994), The Boundary Element Methods in Engineering (2nd ed.), London, etc.: McGraw-Hill, ISBN 978-0-07-707769-3.
- Beer, Gernot; Smith, Ian; Duenser, Christian, The Boundary Element Method with Programming: For Engineers and Scientists, Berlin – Heidelberg – New York: Springer-Verlag, pp. XIV+494, ISBN 978-3-211-71574-1.
- Cheng, Alexander H.-D.; Cheng, Daisy T. (2005), Heritage and early history of the boundary element method, Engineering Analysis with Boundary Elements, 29 (3): 268–302, doi:10.1016/j.enganabound.2004.12.001, Zbl 1182.65005, available also here.
- Chow, Y. L.; Yang, J. J.; Fang, D. G.; Howard, G. E. (March 1991). A closed-form spatial Green's function for the thick microstrip substrate. IEEE Transactions on Microwave Theory and Techniques. 39 (3): 588–592. doi:10.1109/22.75309.
- Ern, A.; Guermond, J.-L. (2004), Theory and Practice of Finite Elements, New York, NY: Springer-Verlag, ISBN 0387205748.
- Fitts, C. R. (2012). Groundwater Science (2nd ed.). San Diego, CA: Elsevier/ Academic Press. ISBN 9780123847058.
- Gibson, Walton C (2008), The Method of Moments in Electromagnetics, Boca Raton, Florida: Chapman & Hall/CRC Press, pp. xv+272, ISBN 978-1-4200-6145-1, MR 2503144, Zbl 1175.78002.
- Gockenbach, M.S. (2006), Understanding and Implementing the Finite Element Method, Philadelphia, PA: SIAM, ISBN 0898716144.
- Grossmann, C.; Roos, H.-G.; Stynes, M. (2007), Numerical Treatment of Partial Differential Equations, Berlin, Germany: Springer-Verlag, ISBN 978-3-540-71584-9.
- Haitjema, H. M. (1995). Analytic element modeling of groundwater flow. San Diego, CA: Academic Press. ISBN 978-0-12-316550-3.
- Haitjema, H. M. (Henk M.) (1995). Analytic element modeling of groundwater flow. San Diego: Academic Press. ISBN 978-0-08-049910-9. OCLC 162129095.
- Hunt, Randall J. (2006). Ground Water Modeling Applications Using the Analytic Element Method. Groundwater. 44 (1): 5–15. doi:10.1111/j.1745-6584.2005.00143.x.
- In electromagnetics, the more traditional term method of moments is often used,

though not always, as a synonymous of boundary element method: see (Walton 2008) for further information on the subject.

- Janković, I.; Fiori, A.; Dagan, G. (2006). Modeling flow and transport in highly heterogeneous three-dimensional aquifers: Ergodicity, Gaussianity, and anomalous behavior—1. Conceptual issues and numerical simulations. Water Resources Research. 42 (6): W06D12. Bibcode:2006WRR....42.6D12J. doi:10.1029/2005WR004734. ISSN 1944-7973.
- Johnson, C. (2009), Numemerical Solution of Partial Differential Equations by the Finite Element Method, Dover, ISBN 978-0486469003.
- Katsikadelis, John T. (2002), Boundary Elements Theory and Applications, Amsterdam: Elsevier, pp. XIV+336, ISBN 978-0-080-44107-8.
- Kolaei, Amir; Rakheja, Subhash; Richard, Marc J. (2014-11-14). Volume 4A: Dynamics, Vibration, and Control. pp. V04AT04A067. doi:10.1115/IMECE2014-37271. ISBN 978-0-7918-4647-6.
- Kolaei, Amir; Rakheja, Subhash; Richard, Marc J. (2015-01-31). A coupled multimodal and boundary-element method for analysis of anti-slosh effectiveness of partial baffles in a partly-filled container. Computers & Fluids. 107: 43–58. doi:10.1016/j.compfluid.2014.10.013.
- Pohrt, R.; Li, Q. (2014-10-01). Complete boundary element formulation for normal and tangential contact problems. Physical Mesomechanics. 17 (4): 334–340. doi:10.1134/S1029959914040109. ISSN 1029-9599.
- Pohrt, Roman; Popov, Valentin L. (2015-04-09). Adhesive contact simulation of elastic solids using local mesh-dependent detachment criterion in boundary elements method. Facta Universitatis, Series: Mechanical Engineering. 13 (1): 3–10.
- Popov, Valentin (2017). Contact Mechanics and Friction - Physical Principles and (Chapter 19). Springer. pp. 337–341. ISBN 9783662530801.
- Strack, O. D. L.; Ausk, B. K. (August 2015). A formulation for vertically integrated groundwater flow in a stratified coastal aquifer: STRATIFIED COASTAL AQUIFER FLOW. Water Resources Research. 51 (8): 6756–6775. doi:10.1002/2015WR016887.

MCQs

1. **How is Assembly of stiffness matrix symbolically denoted?**
   a. K={k}e
   b. K←$\sum_e K^e$
   c. K←$\sum K^e$
   d. Undefined
2. **What is the Strain energy equation?**
   a. $U_e=12q^T k^e q$
   b. $U_e=12qekeq$

c. U_e=12qke

d. U_e=12qTke

3. **What is the actual equation of stiffness matrix?**

a. K=[1−1−11]

b. K=AEl[1−1]

c. K=AEl

d. K=AEl[1−1−11]

4. **From where does the global load vector F is assembled?**

a. Element force vectors only

b. Point loads only

c. Both element force vectors and point loads

d. Undefined

5. **Principal of minimum potential energy follows directly from the principal of _____**

a. Elastic energy

b. Virtual work energy

c. Kinetic energy

d. Potential energy

6. **The points at where kinetic energy increases dramatically then those points are called _____**

a. Stable equilibrium points

b. Unstable equilibrium points

c. Equilibrium points

d. Unique points

7. **We can obtain same assembly procedure by Stiffness matrix method and _____**

a. Potential energy method

b. Rayleigh method

c. Galerkin approach

d. Vector method

8. **By element stiffness matrix we can get relation of members in an object in _____**

a. Different matrices

b. One matrix

c. Identity matrix

d. Singular matrix

9. **What is the Global stiffness method called?**
   a. Multiple matrix
   b. Direct stiffness matrix
   c. Unique matrix
   d. Vector matrix
10. **Which technique do traditional workloads use?**
   a. Scale out technique
   b. Scale up technique
   c. Building technique
   d. Shrinking technique

ANSWERS

1 (b)	2 (a)	3 (d)	4 (c)	5 (b)	6 (b)
7 (c)	8 (b)	9 (b)	10 (b)		

7 Some Biharmonic Problems

OBJECTIVES

After studying this chapter, you will be able to know:

- Case Study: Plane Strain Compression
- Case Study: Stresses in Concentric Cylinders
- Case Study: Stress Concentration Near a Hole in a Flat Plate

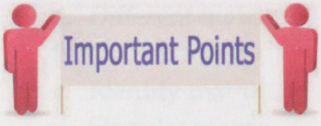
1. INTRODUCTION

In mathematics, the biharmonic equation is a fourth-order partial differential equation which arises in areas of continuum mechanics, including linear elasticity theory and the solution of Stokes flows. Specifically, it is used in the modeling of thin structures that react elastically to external forces.

Plane strain is applicable to rolling, drawing and forging where flow in a particular direction is constrained by the geometry of the machinery, e.g. a well-lubricated die wall. A specific example of this is in rolling, where the major deformation occurs perpendicular to the roll axis. The material becomes thinner and longer but not wider. The use of FEA extends well beyond simple deformation since the approach can handle static and dynamic loading (with time-dependent changes in loads), two or three dimensional objects and different types of loading, e.g. thermal, electromagnetic as well as mechanical forces. Many commercial packages are now available.

2. CASE STUDY: PLANE STRAIN COMPRESSION

Formability is the materials ability to be shaped in to semi-finished and finished products by manufacturing techniques like rolling, forging, extrusion, deep drawing, etc. Many transportation equipment like cars, trucks, air planes, etc., use thin sheet metals generally steel and aluminum alloys formed in to respective shapes by processes such as deep drawing, bending, etc., and joined to each other to make an envelope. With increasing need for light weighting the equipment for fuel efficiency, many sheet metal producers are engaged in material developments like alloy development, coating development, etc., to



With increasing need for light weighting the equipment for fuel efficiency, many sheet metal producers are engaged in material developments like alloy development, coating development, etc.

improve the formability of high strength and light gage products. Both the sheet metal producer and fabricators need to know if a material can be formed in to the shapes required by traditional processes to form the body envelop of a car, truck or air craft, etc.

These operations are done at ambient temperature or warm temperatures, but at high process speeds. This cold formability is very critical for component manufacturing. Equally several finished, semi-finished metal product manufactures like forging, extrusion companies and flat rolling companies would like to understand the formability of materials specifically metals and alloys. These operations are generally done at high temperatures. Thus, hot formability or warm formability of the materials is also important. These are assessed using simple tests like tensile tests, compression test, torsion test, bending test, drawing test, etc.

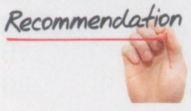
The plane strain compression test is a specialized test used on some materials ranging from metals to soils. One variation of the test is also known as the Watts-Ford test. It is an engineering test, and is a particularly specialized way of determining some of the material characteristics of the metal being tested, and its specialization can be summarized by this quote:

The test is useful when the sheet pieces are too small for a tensile test of a balanced biaxial test. It can give stress-strain curves up to considerably higher strains than tensile tests.

Plane-strain compression testing is typically used for measuring mechanical properties and for exploring microstructure development in the course of thermomechanical treatment. During the test the specimen is placed between the punches and the constrain plates. When the upper punch is pushed down during the material test, the specimen is extended to horizontal directions. Friction between the tool and the specimen can be reduced by applying lubricants, such as graphite, MoS2, glass or PTFE(Teflon).

The testing essentially consists of a thin metal bar being compressed by two equally wide compressive strips, which are located of opposite sides of the thin bar. Then, over a range of increasing loads on the bar, the compressive forces lead to the thickness of the metal bar being reduced. This change of thickness is then measured sequentially after each loading, and after some mathematics a stress-strain curve can be plotted.

The advantages of the Watts-Ford test are that it is convenient for testing thin sheets or strips, it is similar to a rolling process (in manufacturing



Change of thickness is then measured sequentially after each loading, and after some mathematics a stress-strain curve can be plotted.

analyses), frictional effects may be minimized, here is no 'barelling' as would occur in a cylindrical compression test, and the plane strain deformation eases the analysis.

Stress-Strain Curve

The stress-strain curve is the relationship between the stress (force per unit area) and strain (resulting compression/stretching, known as deformation) that a particular material displays; stress–strain curves of various materials differ widely, and different tensile tests conducted on the same material yield different results depending upon the temperature of the specimen and the speed of the loading. When performing Watts-Ford tests, temperatures of the metal specimens will vary from 800-1100 °C and strain rates of (0.01-10 s^{-1}).

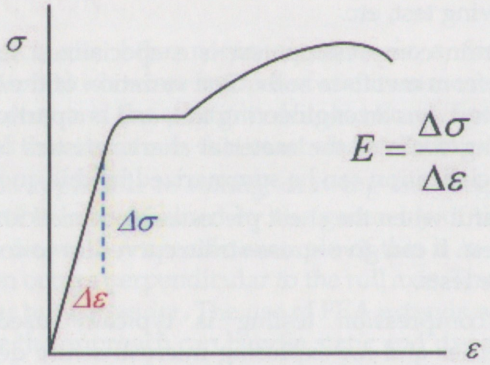


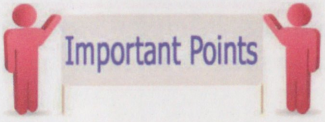
Fig. 7.1 Stress-strain curve

Pressure

The average pressure on a unit of area of the contact surface between the punch and the specimen is expressed as: P= F/(wb), where F is force, w is the punch width, b is the specimen width.

3. CASE STUDY: STRESSES IN CONCENTRIC CYLINDERS

The classical example (and namesake) of hoop stress is the tension applied to the iron bands, or hoops, of a wooden barrel. In a straight, closed pipe, any force applied to the cylindrical pipe wall by a pressure differential will ultimately give rise to hoop stresses. Similarly, if this pipe has flat end caps, any force applied to them by static pressure will induce a perpendicular axial



In a straight, closed pipe, any force applied to the cylindrical pipe wall by a pressure differential will ultimately give rise to hoop stresses.

stress on the same pipe wall. Thin sections often have negligibly small radial stress, but accurate models of thicker-walled cylindrical shells require such stresses to be considered.

In thick-walled pressure vessels, construction techniques allowing for favorable initial stress patterns can be utilized. These compressive stresses at the inner surface reduce the overall hoop stress in pressurized cylinders. Cylindrical vessels of this nature are generally constructed from concentric cylinders shrunk over (or expanded into) one another, i.e., built-up shrink-fit cylinders, but can also be performed to singular cylinders though autofrettage of thick cylinders.

When the concentric cylinders are subjected to either an axial load or a uniform radial stress then there are no shear stresses produced. Further, in an infinite cylinder away from the ends, the stresses are independent of X. For this state of stress, there will be one equilibrium equation given as

$$\frac{d\sigma_{rr}}{dr}+\frac{\sigma_{rr}-\sigma_{\theta\theta}}{r}=0$$

where,σ_{rr} and $\sigma_{\theta\theta}$are radial and circumferential stresses, respectively. For a cylindrical coordinate system with no shear stresses, we can write normal stresses for transversely isotropic material as

$$\sigma_{xx}=C_{11}^{*}\varepsilon_{xx}+C_{12}^{*}\varepsilon_{\theta\theta}+C_{12}^{*}\varepsilon_{rr}$$
$$\sigma_{\theta\theta}=C_{12}^{*}\varepsilon_{xx}+C_{22}^{*}\varepsilon_{\theta\theta}+C_{23}^{*}\varepsilon_{rr}$$
$$\sigma_{rr}=C_{12}^{*}\varepsilon_{xx}+C_{23}^{*}\varepsilon_{\theta\theta}+C_{22}^{*}\varepsilon_{rr}$$

Similarly, for transversely isotropic fibre and isotropic matrix the normal stresses are given as

$$\sigma_{xx}^{(f)}=C_{11}^{f}\varepsilon_{xx}+C_{12}^{f}\varepsilon_{\theta\theta}+C_{12}^{f}\varepsilon_{rr},\quad \sigma_{xx}^{(m)}=C_{11}^{m}\varepsilon_{xx}+C_{12}^{m}\varepsilon_{\theta\theta}+C_{12}^{m}\varepsilon_{rr},$$
$$\sigma_{\theta\theta}^{(f)}=C_{12}^{f}\varepsilon_{xx}+C_{22}^{f}\varepsilon_{\theta\theta}+C_{23}^{f}\varepsilon_{rr},\quad \sigma_{\theta\theta}^{(m)}=C_{12}^{m}\varepsilon_{xx}+C_{11}^{m}\varepsilon_{\theta\theta}+C_{12}^{m}\varepsilon_{rr},$$
$$\sigma_{rr}^{(f)}=C_{12}^{f}\varepsilon_{xx}+C_{23}^{f}\varepsilon_{\theta\theta}+C_{22}^{f}\varepsilon_{rr},\quad \sigma_{rr}^{(m)}=C_{12}^{m}\varepsilon_{xx}+C_{12}^{m}\varepsilon_{\theta\theta}+C_{11}^{m}\varepsilon_{rr},$$

Now, we write the strain displacement relations in cylindrical coordinates as

$$\varepsilon_{xx}=\frac{\partial u}{\partial x}\qquad \varepsilon_{\theta\theta}=\frac{1\partial v}{r\partial\theta}+\frac{w}{r'}\qquad \varepsilon_{rr}=\frac{\partial w}{\partial r}$$
$$\gamma_{\theta r}=\frac{1}{r}\left(\frac{\partial w}{\partial\theta}-v\,r\frac{\partial v}{\partial r}\right),\qquad \gamma_{xr}=\frac{\partial u}{\partial r}+\frac{\partial w}{\partial x'}\qquad \gamma_{x\theta}=\frac{\partial w}{\partial x}+\frac{1\partial u}{r\partial\theta'}$$

where, u,v and w are axial, circumferential or tangential and radial displacements, respectively. Further, with no shear effects and axisymmetry the tangential displacement is zero. It should be noted that the stresses are not

function of x. Hence, the strains are independent ofx. Hence, the displacement W can be a function of r. However, u can be function of r and, at most, a linear function of x. Hence, in the expression for strain ε_{xx} there is partial derivative and not in the expression for ε_{rr}. For the expression of ε_{rr} we have the full derivative with respect to r.

Thus, the normal strains for the displacements in this case simplify to

$$\varepsilon_{xx} = \frac{\partial u}{\partial x}, \varepsilon_{\theta\theta} = \frac{w}{r}, \varepsilon_{rr} = \frac{dw}{dr},$$

When these strain-displacement relations are used in stress-strain relations then the equilibrium equation in Equation gives an ordinary differential equation in W as

$$\frac{d^2w}{dr^2} + \frac{1}{r}\frac{dw}{dr} - \frac{w}{r^2} = 0$$

Practical Effects

Engineering

Fracture is governed by the hoop stress in the absence of other external loads since it is the largest principal stress. Note that a hoop experiences the greatest stress at its inside (the outside and inside experience the same total strain, which is distributed over different circumferences); hence cracks in pipes should theoretically start from inside the pipe. This is why pipe inspections after earthquakes usually involve sending a camera inside a pipe to inspect for cracks. Yielding is governed by an equivalent stress that includes hoop stress and the longitudinal or radial stress when absent.

Medicine

In the pathology of vascular or gastrointestinal walls, the wall tension represents the muscular tension on the wall of the vessel. As a result of the Law of Laplace, if an aneurysm forms in a blood vessel wall, the radius of the vessel has increased. This means that the inward force on the vessel decreases, and therefore the aneurysm will continue to expand until it ruptures. A similar logic applies to the formation of diverticuli in the gut.

The Concept of Continuum

Continuum mechanics is a branch of mechanics that deals with the mechanical behavior of materials modeled as a continuous mass rather than



Note that a hoop experiences the greatest stress at its inside (the outside and inside experience the same total strain, which is distributed over different circumferences).

as discrete particles. The French mathematician Augustin-Louis Cauchy was the first to formulate such models in the 19th century.

Modeling an object as a continuum assumes that the substance of the object completely fills the space it occupies. Modeling objects in this way ignores the fact that matter is made of atoms, and so is not continuous; however, on length scales much greater than that of inter-atomic distances, such models are highly accurate. Fundamental physical laws such as the conservation of mass, the conservation of momentum, and the conservation of energy may be applied to such models to derive differential equations describing the behavior of such objects, and some information about the material under investigation is added through constitutive relations.

Continuum mechanics deals with physical properties of solids and fluids which are independent of any particular coordinate system in which they are observed. These physical properties are then represented by tensors, which are mathematical objects that have the required property of being independent of coordinate system. These tensors can be expressed in coordinate systems for computational convenience.

Materials, such as solids, liquids and gases, are composed of molecules separated by space. On a microscopic scale, materials have cracks and discontinuities. However, certain physical phenomena can be modeled assuming the materials exist as a continuum, meaning the matter in the body is continuously distributed and fills the entire region of space it occupies. A continuum is a body that can be continually sub-divided into infinitesimal elements with properties being those of the bulk material.

The validity of the continuum assumption may be verified by a theoretical analysis, in which either some clear periodicity is identified or statistical homogeneity and ergodicity of the microstructure exists. More specifically, the continuum hypothesis/assumption hinges on the concepts of a representative elementary volume and separation of scales based on the Hill–Mandel condition. This condition provides a link between an experimentalist's and a theoretician's viewpoint on constitutive equations (linear and nonlinear elastic/inelastic or coupled fields) as well as a way of spatial and statistical averaging of the microstructure.

When the separation of scales does not hold, or when one wants to establish a continuum of a finer resolution than that of the representative volume element (RVE) size, one employs a statistical volume element (SVE), which, in turn, leads to random continuum fields.

The latter then provide a micromechanics basis for stochastic finite elements (SFE). The levels of SVE and RVE link continuum mechanics to statistical mechanics. The RVE may be assessed only in a limited way via

experimental testing: when the constitutive response becomes spatially homogeneous.

Specifically for fluids, the Knudsen number is used to assess to what extent the approximation of continuity can be made.

Car Traffic as an Introductory Example

Consider car traffic on a highway, with just one lane for simplicity. Somewhat surprisingly, and in a tribute to its effectiveness, continuum mechanics effectively models the movement of cars via a partial differential equation (PDE) for the density of cars. The familiarity of this situation empowers us to understand a little of the continuum-discrete dichotomy underlying continuum modelling in general.

To start modelling define that: x measures distance (in km) along the highway; t is time (in minutes); ρ(x, t) is the density of cars on the highway (in cars/km in the lane); and u (x, t) is the flow velocity (average velocity) of those cars 'at' position x.

Conservation Derives a PDE (Partial Differential Equation)

Cars do not appear and disappear. Consider any group of cars: from the particular car at the back of the group located at x = a(t) to the particular car at the front located at x = b(t). The total number of cars in this group $N = \int_{a(t)}^{b(t)} \rho(x,t)dx$. Since cars are conserved (if there is overtaking, then the `car at the front/back' may become a different car) dN/dt = 0. But via the Leibniz integral rule

$$\begin{aligned}\frac{dN}{dt} &= \frac{d}{dt}\int_{a(t)}^{b(t)} \rho(x,t)dx \\ &= \int_a^b \frac{\partial \rho}{\partial t}dx + \rho(b,t)\frac{db}{dt} - \rho(a,t)\frac{da}{dt} \\ &= \int_a^b \frac{\partial \rho}{\partial t}dx + \rho(b,t)u(b,t) - \rho(a,t)u(a,t) \\ &= \int_a^b \left[\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x}(\rho u)\right]dx\end{aligned}$$

IMPORTANT

Somewhat surprisingly, and in a tribute to its effectiveness, continuum mechanics effectively models the movement of cars via a partial differential equation (PDE) for the density of cars.

4. CASE STUDY: STRESS CONCENTRATION NEAR A HOLE IN A FLAT PLATE

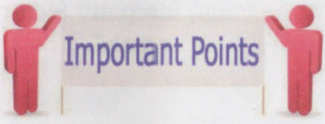
Using finite element method, models of different element mesh density for the center hole in flat plate were established according to one actual steel plate with central circular hole. The stress concentration coefficients of these models were calculated by means of numerical method. Then the calculated results were compared with the theoretical solution results of Elastic Mechanics. And it was analyzed subsequently that the influences which were caused by different element mesh density on the stress concentration coefficients around the hole in flat plate.

The relationship between the element mesh density and the stress concentration coefficients around the hole in flat plate was proposed finally. Results show that the element mesh density for calculation affects the stress concentration coefficients around the hole in flat plate significantly, but the influence is few when the mesh density increases to a certain extent.

The story of the analysis of stress concentrations begins in 1898 with Ernst Gustav Kirsch's linear elastic solution for stresses around a hole in an infinite plate. Kirsch's solution contains the well-known factor-of-three stress concentration at the hole under uniaxial loading. But there is more to the story. The stress concentration can in fact vary from two to four for more complex loading conditions, i.e., stress states different from uniaxial tension. We begin by first reviewing Kirsch's solution for the basic case of uniaxial tension, and then use superposition techniques to develop solutions for the more complex cases of equibiaxial tension and shear. This is followed by the solution for general 2-D loading. And the page concludes with stress concentrations in finite width plates.

Stress Concentration Factors Calculators to calculate theoretical stress concentration factors (K_t) in elastic range for different type of stress raisers and loading conditions (tension, bending and torsion).

Dimensional changes and discontinuities of a member in a loaded structure causes variations of stress and high stresses concentrate near these dimensional changes. This situation of high stresses near dimensional changes and discontinuities of a member (holes, sharp corners, cracks etc.) is called stress concentration. The ratio of peak stress near stress riser to average stress over the member is called stress concentration factor.



The stress concentration coefficients of these models were calculated by means of numerical method.

The degree of concentration of a discontinuity under typically tensile loads can be expressed as a non-dimensional stress concentration factor K_t, which is the ratio of the highest stress to the nominal far field stress. For a circular hole in an infinite plate, $K_t = 3$. The stress concentration factor should not be confused with the stress intensity factor, which is used to define the effect of a crack on the stresses in the region around a crack tip.

For ductile materials, large loads can cause localised plastic deformation or yielding that will typically occur first at a stress concentration allowing a redistribution of stress and enabling the component to continue to carry load. Brittle materials will typically fail at the stress concentration. However, repeated low level loading may cause a fatigue crack to initiate and slowly grow at a stress concentration leading to the failure of even ductile materials. Fatigue cracks always start at stress raisers, so removing such defects increases the fatigue strength.

Stress Concentration Factor

The stress concentration factor, K_t, is the ratio of the highest stress σ_{max} to a nominal stress σ_{nom} of the gross cross-section and defined as

$$K_t = \frac{\sigma_{max}}{\sigma_{nom}}$$

Note that the dimensionless stress concentration factor is a function of the geometry shape and independent of its size. These factors can be found in typical engineering reference materials.

E. Kirsch derived the equations for the elastic stress distribution around a hole. The maximum stress felt near a hole or notch occurs in the area of lowest radius of curvature. In an elliptical hole of length 2a and width 2b, under a nominal or far-field stress σ_{nom}, the stress at the ends of the major axes is given by Inglis' equation:

$$\sigma_{max} = \sigma_{nom}\left(1+2\frac{a}{b}\right) = \sigma\left(1+2\sqrt{\frac{a}{\rho}}\right)$$

where ρ is the radius of curvature of the elliptical hole. For circular holes in an infinite plate where a = b, the stress concentration factor is $K_t = 3$.

As the radius of curvature approaches zero, such as at the tip of a sharp crack, the maximum stress approaches infinity and a stress concentration factor cannot therefore be used for a crack. Instead, the stress intensity factor which defines the scaling of the stress field around a crack tip, is used.

Limiting the Effects of Stress Concentrations

Known as crack tip blunting, a counter-intuitive method of reducing one of the worst types of stress concentrations, a crack, is to drill a large hole at the end of the crack. The drilled hole, with its relatively large size, serves to increase the effective crack tip radius and thus reduce the stress concentration.

Another method used to decrease the stress concentration is by adding a fillet to internal corners. This reduces the stress concentration and results in smoother flow of stress streamlines.

In a threaded component, the force flow line is bent as it passes from shank portion to threaded portion; as a result, stress concentration takes place. To reduce this, a small undercut is made between the shank and threaded portions.

Stress Concentration Factors for Flat Plates and Cylinders

In a structure or machine part having a notch or any abrupt change in cross section, the maximum stress will occur at this location and will be greater than the stress calculated by elementary formulas based upon simplified assumptions as to the stress distribution. The ratio of this maximum stress to the nominal stress (calculated by the elementary formulas) is the stress-concentration factor K_t. This is a constant for the particular geometry and is independent of the material, provided it is isotropic. The stress-concentration factor may be determined experimentally or, in some cases, theoretically from the mathematical theory of elasticity.

The factors shown on this webpage were determined from both photoelastic tests and the theory of elasticity. Stress concentration will cause failure of brittle materials if the concentrated stress is larger than the ultimate strength of the material. In ductile materials, concentrated stresses higher than the yield strength will generally cause local plastic deformation and redistribution of stresses (rendering them more uniform). On the other hand, even with ductile materials areas of stress concentration are possible sites for fatigue if the component is cyclically loaded.

Simplifying FEA Models: Plane Stress and Plane Strain

The two FEA methods are called plane stress and plane strain. Both use 2D planar elements that look like thin shell elements and are meshed using planar surface geometry.

IMPORTANT

The ratio of this maximum stress to the nominal stress is the stress-concentration factor Kt.

Plane Stress Analysis

Figure (below) shows the important facts about plane stress analysis. The structural region is assumed to lie in the 2D xy plane, with the third structural dimension relatively small. In the figure, this is the thickness in the z direction. Stresses exist in the 2D plane as sigma x, sigma y (direct stresses) and sigma xy (in-plane shear stress). Each of these stresses is constant through the thickness as shown in the inset. In addition there can be no stress in the z direction. This stress-strain material relationship is defined in 2D plane stress elements used in this type of analysis.

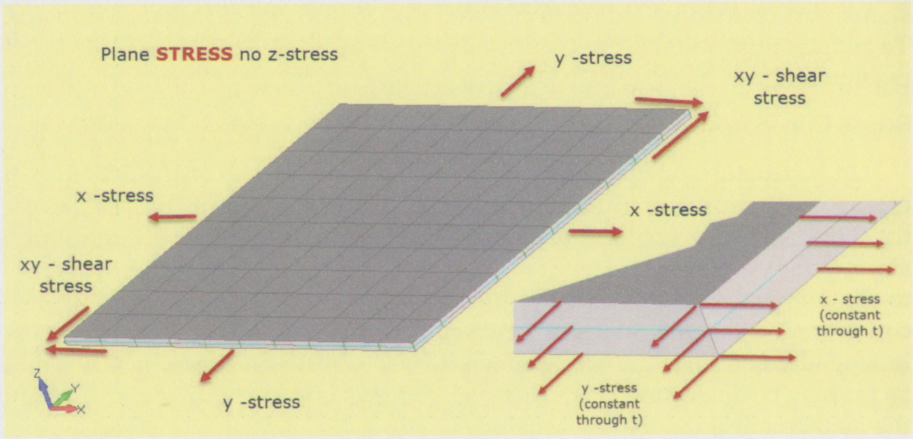


Fig. 7.2 Plane stress; stress state assumptions.

The lack of z stress is the way to remember the element type designation plane stress (i.e. only in-plane stresses allowed). There are also no through thickness shear stresses. We could load the plane stress model in Figure (above) with a bi-axial load and calculate sigma x and sigma y. There is no sigma z. We can also calculate the corresponding in-plane strains e-x and e-y. If we apply a shear load, we can find shear stress sigma xy and shear strain e-xy. Interestingly, we can also calculate the through thickness strain e-z. This is not zero, as the model is free to thin down in z as it stretches in x and y. There is no out-of-plane constraint to prevent this. In some solvers we can recover the through thickness strain e-z and use it to calculate the change in thickness. (If the model is in bi-axial compression then the z section will get thicker). This is usually considered a secondary strain and may not be available for output—but it is there and can be calculated manually if needed.

Figure (below) shows an aluminum lug component. The lug protrudes from a composite sheet layup which has plies positioned and bonded over the tangs (or legs) and lower body section. The tangs transfer the load applied to

the lug into the composite structure. In practice, the plies would be stepped to allow a smooth shear transfer through the bond into the composite. The shear transfer into the composite is simulated here by diffused surface traction forces "pulling" on the tangs. These balance the applied lug load.

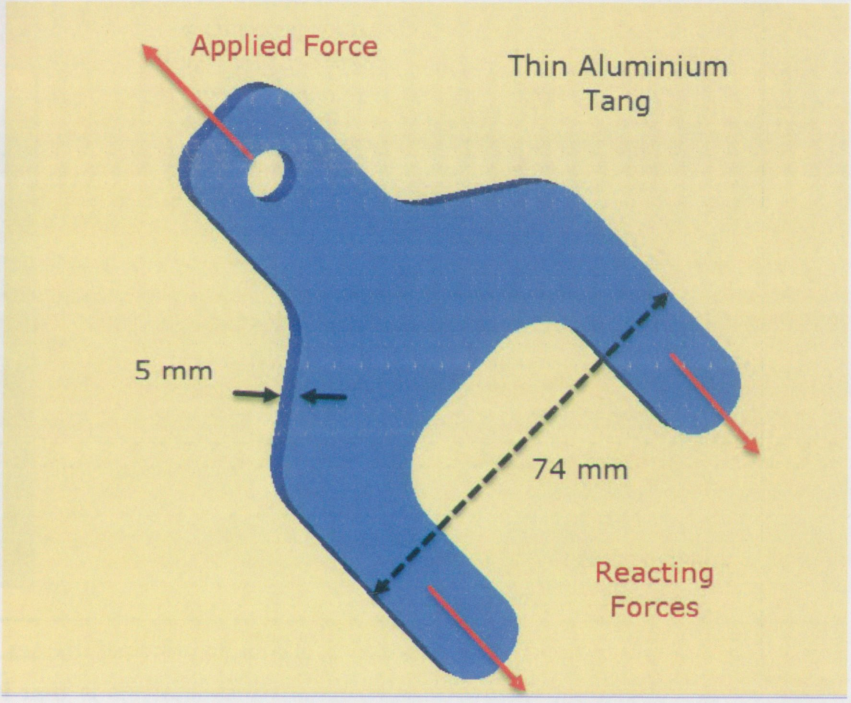


Fig. 7.3 Thin-walled Aluminum tang transferring load into composite structure.

The key assumption here is that through thickness stresses are zero and the in-plane stresses are constant through thickness in the component. This means the local detail of the shear load transfer from composite to tang is poorly modeled. However the focus of this analysis is to check sizing of lug and tang cross section clear of the composite, using in-plane stresses.

The thickness of the component is small compared to other dimensions. This value is input as the actual thickness in the plane stress element definition.

Figure (below) shows the FEA model and the calculated maximum principal stresses. The areas of interest are around the lug and the shoulder radii. In the real world the stress state at the stress concentrations would be 3D and through thickness sigma z stresses and shear stresses would balance locally. However, it is very reasonable here to assume the in-plane stresses

dominate. This is the same assumption implicit in most traditional stress concentration (K_t) calculations found in handbooks.

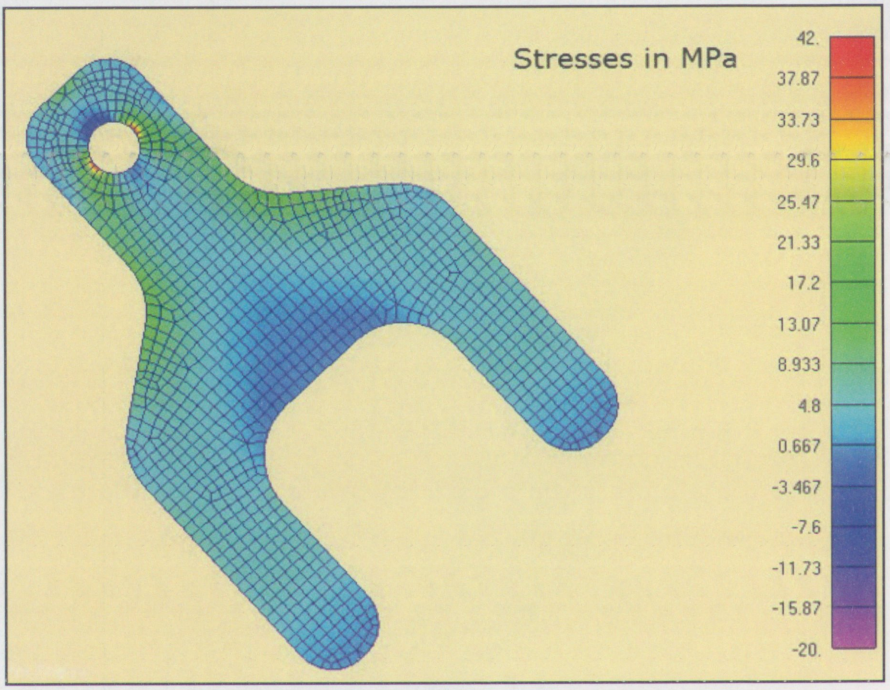


Fig. 7.4 2D Plane stress elements showing maximum principal stresses

One of the convenient features of the plane stress analysis is that it is a strictly 2D analysis, so only three degrees of freedom (DOF) have to be constrained (in-plane translations x, y and rotation about z axis). This lends itself to the 3-2-1 minimum constraint method with balanced load. In a 2D case this degenerates to a 2-1 method. One node has DOF x and y constrained, a second appropriate orthogonal node has DOF x constrained. This allows the reaction load in the tangs to be applied directly as diffused balancing loads. It would be difficult to simulate this boundary condition via constraints to ground.

The through thickness e-z strain and hence thinning of the tangs could be calculated as a secondary effect.

Plane Strain Analysis

Figure (below) shows the essence of the plane strain method. Again, 2D planar elements are used, but with subtly different assumptions. The in-plane stresses x, y and xy are developed as before. However this time it is the out-of-

plane, or through thickness z strain which is set to zero. So plane strain analysis only allows strains in-plane. This works well to represent thick structures such as shown. The presence of this much material tends to stabilize the component and prevent it straining in z. This also means that constant through thickness z stresses are developed in the structure. This stress-strain material relationship is defined in 2D plane strain elements used in this type of analysis.

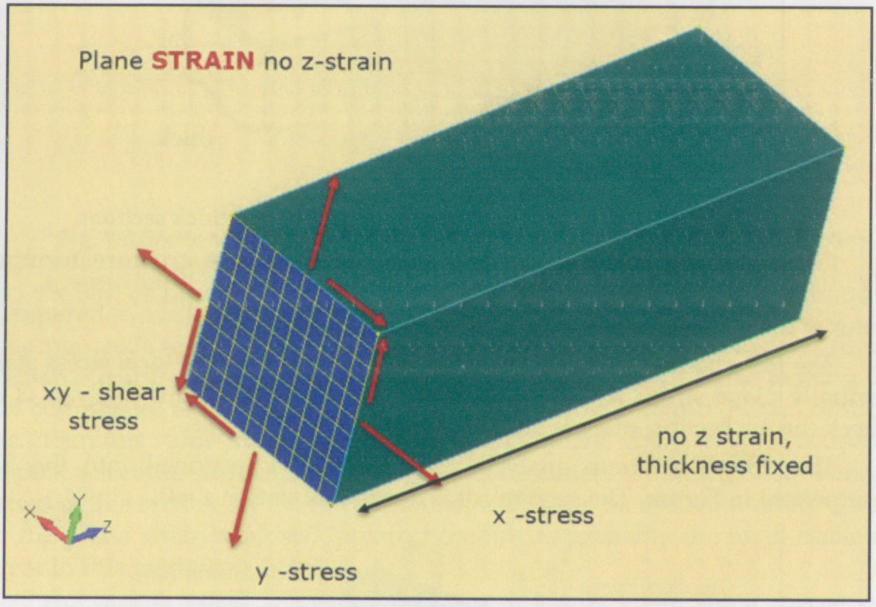


Fig. 7.5 Plane strain analysis; stress and strain state assumptions.

The figure shows the orientation of the 2D plane strain elements as a cut section through a typical deep component. The assumption is that the stress state at this cut section will be duplicated at any xy plane cut (z station) through the component. The component is assumed to be prismatic (having a constant cross section) down its length.

In practice we use this method where the stress state is varying slowly from plane to plane in a deep component. There should be enough material through depth to stabilize and eliminate the through thickness strain. This is the same principle used on fracture toughness specimens shown in Figure.

A failure under plane strain conditions is shown for the center section of the thick specimen. The failure at the free edges and the thin section is a different mode, more like a plane stress state. A plane strain FEA model would by definition be a good representation of the centerline thick specimen behavior, but not of the free edges or the thin specimen.

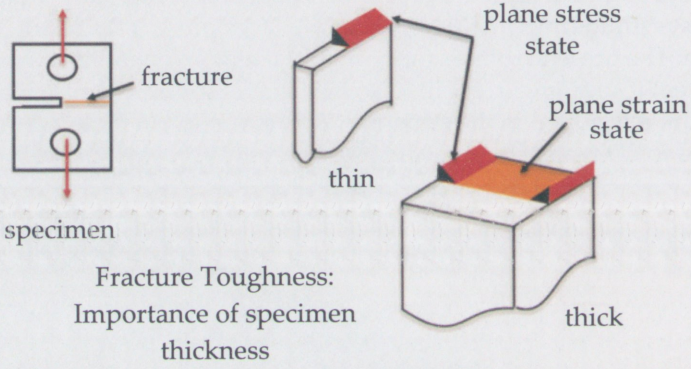


Fig. 7.6 Fracture toughness specimens; thin and thick sections.

Figure shows another component used in a composite structure, forming a bonded joint. Here the section is constant and deep enough so that we can assume the stresses are also constant with depth.

The free end surface faces (at +z, -z) will have a different local stress state (actually plane stress, as noted), however the objective of this analysis is to check the net section stresses on the centerline (z = 0).

The 2D plane strain analysis mesh is shown sectioned into the 3D component in Figure. The section cut is defined at station z = 0.

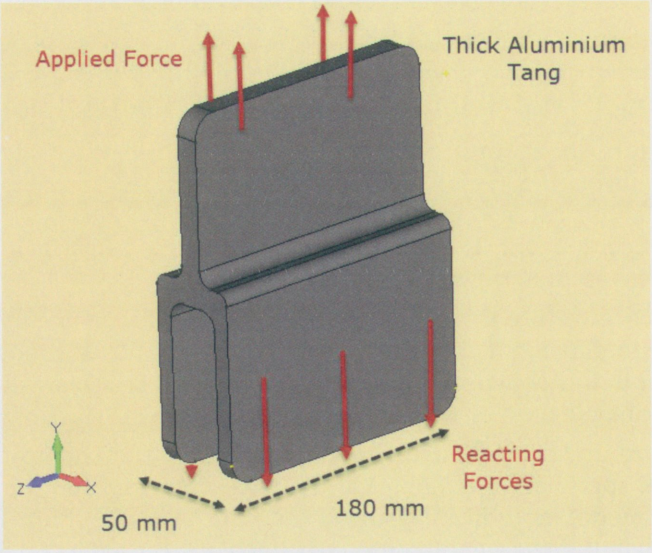


Fig. 7.7 Deep-section aluminum tang.

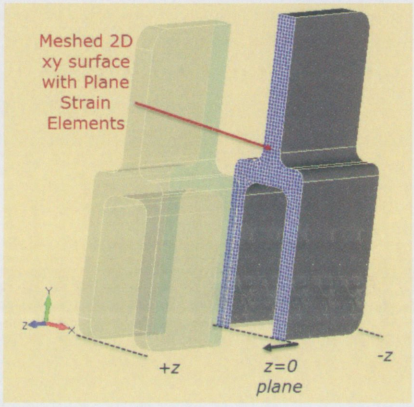


Fig. 7.8 Section cut through solid section to develop 2D plane strain section.

A very fine 2D plane strain mesh can be used, which will run very quickly compared to a full 3D model. The 2-1 constraint method is used as before. The loading needs to be considered carefully. The "thickness" of the plane strain section is quite arbitrary, and is usually set at 1.0 by default. If the loading on the component is calculated as a running load through depth (N/m, Lbf/inch etc.) then this value can be used directly on the plane strain mesh. It is useful to pick a section, such as the single tang and estimate the nominal or average stress in this section for the full component. This can be used as a sanity check in the plane strain analysis. Incorrect loading is probably the main cause of error in this method.

The results of the analysis are shown in Figure 8 and show clearly the regions of high stress around the shoulder fillet. The stresses are valid for the central depth region of the component (z = 0).

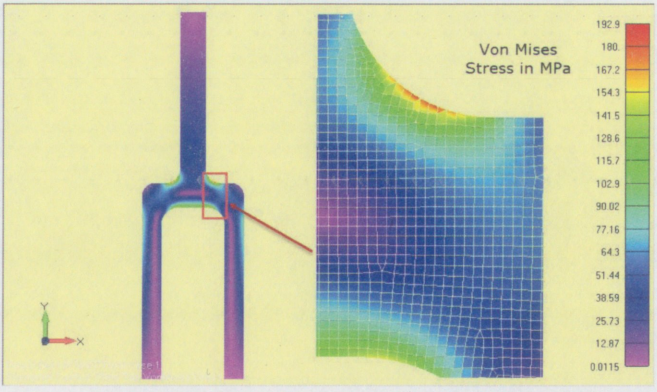


Fig. 7.9 Plane strain analysis results of a deep tang component.

The stress quantities used will depend on the solver used. Some solvers ignore the z direction stresses as secondary and recover the in-plane stresses. The principal stresses and von Mises stresses then relate to a 2D in-plane stress state. If the z direction stress is recovered then it should be clearly identified, so that the 2D in-plane stress state in the x-y plane can be identified.

What exactly does the z direction stress represent? It is the stress developed due to the enforcement of zero z direction strain. The stress acts as if the free end faces of the prismatic section were fixed. At the central plane of a deep section component these will be the complementary stresses needed to hold the zero z direction strain state. In reality as we move toward the free surface faces, the z-stress drops to zero and becomes a plane stress distribution (as seen in the thick fracture mechanics specimen).

In many cases, such as a pressurized cylinder, the end faces are capped and will in fact develop an axial stress due to axial forces. This will be a different stress from the induced axial stress in the plane strain analysis. A hand calculation will be needed to calculate the axial stresses, or possible a supplementary axisymmetric model for pressure vessels.

The ease of geometry and mesh construction lends itself well to "what-if" studies or more formal shape optimization studies.

For comparison, a half symmetry full 3D analysis of the deep tang component was done and the results are shown in Fig. 9. The nominal stress across the upper single tang leg is identical in both cases—remember this is the basis of any sanity check.

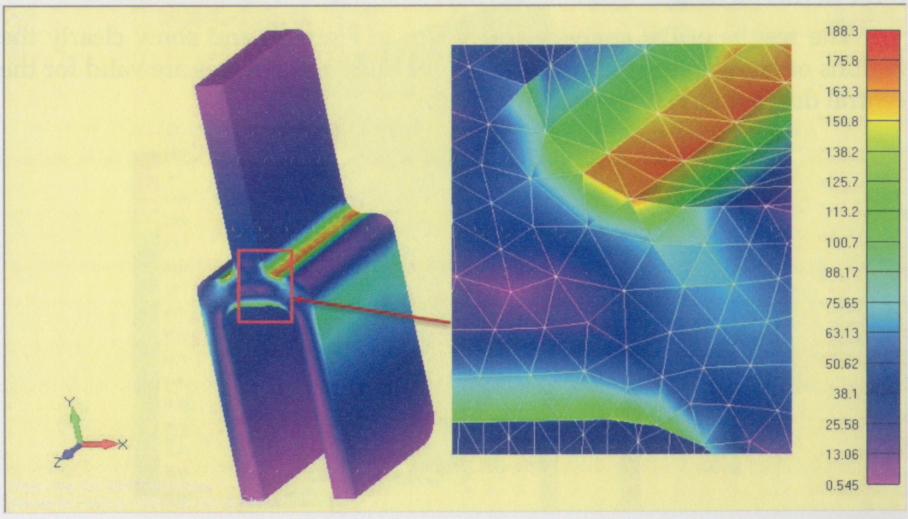


Fig. 7.10 Full 3D model of the deep tang, showing stress results.

The local shoulder stresses are lower by a small percentage in the full model. This is for three reasons. First, the relatively coarse 3D tet mesh is inferior to the very fine 2D plane strain local mesh. A convergence check on the 3D model has not been carried out.

Secondly, there is a small change in geometry at the free surfaces (+z, -z) compared to the z = 0 section due to the end fillets. In this case, the effect is negligible as the fillets are away from the shoulder regions. In many components, however, there will be local fillets, and run out details. which will vary the geometry from a simple xy planar face. Local stress variations at free end faces may have to estimated or checked with a full 3D model.

Finally, the plane strain assumption of a fixed z constraint at +z,-z sections is not true for a finite depth component. The z stress will diffuse to zero at the "real" free faces. This effect is shown in Figure, which uses the 3D model as is and also as a simulation of the plane strain z stress.

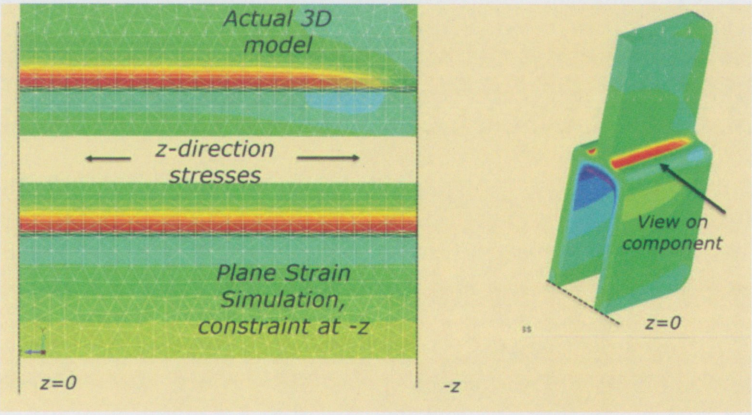


Fig. 7.11 Diffusion of z-stress toward the free surface.

Fast and Efficient

Plane stress and plane strain analyses are useful 2D methods that can often supplement full-scale 3D models. Not all features can be represented, but with some ingenuity, stresses in key areas can at least be estimated. The motivation for using the methods is to enable fast efficient analysis with easy 2D geometry and mesh construction.

CONCLUSION

The plane strain compression test can be scaled with full similarity. Micro structural effects, such as the Hall-Petch effect can be measured with aid of plane strain compression testing of small specimen. The surface roughness

after the compression of the small PSC specimen is influenced by the grain size diameter to specimen thickness ratio and the topography of the roughness correlates with the micro structure.

Comparison of physical experiments with numerical simulations have shown that the production of micro strips by the flat rolling of thin round wire is influenced by size effects. It is most likely that the decreasing yield stress and the increasing friction with decreasing specimen size are the main contributors to those influences.

REFERENCES

- Bender, C.M. and Orzag, S.A. (1978) Advanced Mathematical Methods for Scientists and Engineers, McGraw-Hill, New York.
- Boas, M.L. (1966) Mathematical Methods in the Physical Sciences, 2nd edition, John Wiley, New York.
- Courant, R. and Hilbert, D. (1962) Methods of Mathematical Physics,Vols.1 and 2, Interscience, New York.
- Dennery, P. and Krzywicki, A. (1967) Mathematics for Physicists, Harper and Row, New York.
- Dettman, J.W. (1988) Mathematical Methods in Physics and Engineering, Dover Publications, New York.
- Greenberg, M.D. (1978) Foundations of Applied Mathematics, Prentice-Hall, Englewood Cliffs, N.J.
- Greenberg, M.D. (1998) Advanced Engineering Mathematics, Prentice-Hall, Englewood Cliffs, N.J.
- Haug, E. and Choi, K.K. (1993) Methods of Engineering Mathematics, Prentice-Hall, Englewood Cliffs, N.J.
- Keener, J.P. (1988) Principles of Applied Mathematics, Addison Wesley, New York.
- Kraut, E.A. (1967) Fundamentals of Mathematical Physics, McGraw-Hill, New York.
- Kreyszig, E. (1992) Advanced Engineering Mathematics, John Wiley, New York.
- O'Neil, P.V. (1995) Advanced Engineering Mathematics, Brooks/Cole Publishing Company, Boston.
- Pipes, L.A. and Harvill, L.R. (1970) Applied Mathematics for Engineers and Physicists, McGraw-Hill, New York.

MCQs

1. **For a partial differential equation, in a function φ (x, y) and two variables x, y, what is the form obtained after separation of variables is applied?**

   a. Φ (x, y) = X(x)+Y(y)

   b. Φ (x, y) = X(x)-Y(y)

c. $\Phi(x, y) = X(x)Y(y)$

d. $\Phi(x, y) = X(x)/Y(y)$

2. **The symbol used for partial derivatives, ∂, was first used in mathematics by Marquis de Condorcet.**

   a. True

   b. False

3. **Separation of variables was first used by L'Hospital in 1750.**

   a. False

   b. True

4. **A particular solution for an equation is derived by eliminating arbitrary constants.**

   a. True

   b. False

5. **Separation of variables, in Mathematics, is also known as Fourier method.**

   a. False

   b. True

6. **Which of the following equations cannot be solved by using the method of Separation of variables?**

   a. Laplace Equation

   b. Helmholtz Equation

   c. Alpha Equation

   d. Biharmonic Equation

7. **In the given figure a stepped column carries loads. What will be the maximum normal stress in the column at B in the larger diameter column if the ratio of P/A here is unity?**

   a. 1/1.5

   b. 1

   c. 2/1.5

   d. 2

8. **The stress which acts in a direction perpendicular to the area is called**

   _____

   a. Shear stress

   b. Normal stress

c. Thermal stress

d. None of the mentioned

9. **Which of these are types of Normal stresses?**

a. Tensile and compressive stresses

b. Tensile and thermal stresses

c. Shear and bending

d. Compressive and plane stresses

10. **In a body loaded under plane stress conditions, what is the number of Independent stress components?**

a. 1

b. 2

c. 3

d. 6

11. **If a bar of large length when held vertically and subjected to a load at its lower end, its won-weight produces additional stress. The maximum stress will be _____**

a. At the lower cross-section

b. At the built-in upper cross-section

c. At the central cross-section

d. At every point of the bar

12. **Which type of stress does in a Reinforcement bar is taken by the concrete?**

a. Tensile stress

b. Compressive stress

c. Shear stress

d. Bending stress

ANSWERS

1 (c)	2 (a)	3 (b)	4 (b)	5 (b)	6 (c)
7 (c)	8 (b)	9 (a)	10 (c)	11 (b)	12 (b)

8 Further Applications

OBJECTIVES

After studying this chapter, you will be able to know:

- Axi-Symmetric Problems
- Biharmonic Problems Involving Incompressible Materials
- Plate and Shell Problems
- Isoparametric Elements
- Nonlinear Problems

1. INTRODUCTION

Axisymmetric problems occur in a large number of different branches of industry and take the form of circular cylinders, cones, domes, toroids, etc. In practical terms, they appear as cooling towers, submarine pressure hulls, offshore drilling rigs, aerosol cans, radomes, nuclear reactors, etc. This chapter focuses on both structural and steady-state field problems. It explains how two-dimensional axisymmetric field problems can be solved with the aid of the in-plane annuli elements.

Each annular element is governed by two nodal circles, one internal and one external. The chapter demonstrates how three-dimensional axisymmetric field problems can be analyzed with the aid of "solid" rings, the cross-sections of which can be of triangular or quadrilateral shape. Because of the axisymmetric nature of such problems, the mathematical formulation can follow similar lines to that adopted for in-plane problems.

2. AXI-SYMMETRIC PROBLEMS

The finite element method (FEM) is a powerful technique originally developed for numerical solution of complex problems in structural mechanics, and it remains the method of choice for complex systems. In the FEM, the structural system is modeled by a set of appropriate finite elements interconnected at discrete points called nodes. Elements may have physical properties such as thickness, coefficient of thermal expansion, density, Young's modulus, shear modulus and Poisson's ratio.

The problems in which the body possesses an axis of symmetry and the boundary conditions and other parameters of the problem are symmetric with respect to this axis are called as axisymmetric problems. The cylindrical

coordinate system (r,θ,z) is found to be convenient for analyzing such problems. In this coordinate system, the primary and secondary variables (or their components if they happen to be vectors and tensors) become independent of θ, in axisymmetric problems. Therefore, the domain of analysis, in axisymmetric problems, becomes just a r - z plane. Thus, axisymmetric problems are two-dimensional problems, in some sense but not completely. Further, if primary variable is a vector, its θ-component becomes zero in axisymmetric problems. Because of this, and the fact that r and z components are independent of θ, some of the components of the secondary variable (which happens to be a tensor in this case) become zero. Thus, the size of the problem reduces.

This lecture describes the integral formulation of a typical axisymmetric problem. Its finite element formulation would be similar to 2-D problems, and hence, will not be discussed.

Practical Considerations

From the application point of view, it is important to model the system such that:

- Symmetry or anti-symmetry conditions are exploited in order to reduce the size of the model.
- Displacement compatibility, including any required discontinuity, is ensured at the nodes, and preferably, along the element edges as well, particularly when adjacent elements are of different types, material or thickness. Compatibility of displacements of many nodes can usually be imposed via constraint relations.
- Elements' behaviours must capture the dominant actions of the actual system, both locally and globally.
- The element mesh should be sufficiently fine in order to produce acceptable accuracy. To assess accuracy, the mesh is refined until the important results shows little change. For higher accuracy, the aspect ratio of the elements should be as close to unity as possible, and smaller elements are used over the parts of higher stress gradient.
- Proper support constraints are imposed with special attention paid to nodes on symmetry axes.

Large scale commercial software packages often provide facilities for generating the mesh, and the graphical display of input and output, which greatly facilitate the verification of both input data and interpretation of the results.

The origin of finite method can be traced to the matrix analysis of structures where the concept of a displacement or stiffness matrix approach was introduced. Finite element concepts were developed based on engineering

methods in 1950s. The finite element method obtained its real impetus in the 1960s and 1970s by John Argyris, and co-workers; at the University of Stuttgart, by Ray W. Clough; at the University of California, Berkeley, by Olgierd Zienkiewicz, and co-workers Ernest Hinton, Bruce Irons; at the University of Swansea, by Philippe G. Ciarlet; at the University of Paris; at Cornell University, by Richard Gallagher and co-workers. The original works such as those by Argyris and Clough became the foundation for today's finite element structural analysis methods.

Straight or curved one-dimensional elements with physical properties such as axial, bending, and torsional stiffnesses. This type of element is suitable for modeling cables, braces, trusses, beams, stiffeners, grids and frames. Straight elements usually have two nodes, one at each end, while curved elements will need at least three nodes including the end-nodes. The elements are positioned at the centroidal axis of the actual members.

- Two-dimensional elements that resist only in-plane forces by membrane action (plane stress, plane strain), and plates that resist transverse loads by transverse shear and bending action (plates and shells). They may have a variety of shapes such as flat or curved triangles and quadrilaterals. Nodes are usually placed at the element corners, and if needed for higher accuracy, additional nodes can be placed along the element edges or even within the element. The elements are positioned at the mid-surface of the actual layer thickness.
- Torus-shaped elements for axisymmetric problems such as membranes, thick plates, shells, and solids. The cross-section of the elements are similar to the previously described types: one-dimensional for thin plates and shells, and two-dimensional for solids, thick plates and shells.
- Three-dimensional elements for modeling 3-D solids such as machine components, dams, embankments or soil masses. Common element shapes include tetrahedrals and hexahedrals. Nodes are placed at the vertexes and possibly in the element faces or within the element.

Element Interconnection and Displacement

The elements are interconnected only at the exterior nodes, and altogether they should cover the entire domain as accurately as possible. Nodes will have nodal (vector) displacements or degrees of freedom which may include translations, rotations, and for special applications, higher order derivatives of displacements. When the nodes displace, they will drag the elements along in a certain manner dictated by the element formulation. In other words, displacements of any points in the element will be interpolated from the nodal displacements, and this is the main reason for the approximate nature of the solution.

Theoretical Overview of FEM-Displacement Formulation: From Elements, to System, to Solution

While the theory of FEM can be presented in different perspectives or emphases, its development for structural analysis follows the more traditional approach via the virtual work principle or the minimum total potential energy principle. The virtual work principle approach is more general as it is applicable to both linear and non-linear material behaviours. The virtual work method is an expression of conservation of energy: for conservative systems, the work added to the system by a set of applied forces is equal to the energy stored in the system in the form of strain energy of the structure's components.

The principle of virtual displacements for the structural system expresses the mathematical identity of external and internal virtual work:

External virtual work = $\int_v = \delta\epsilon^T \sigma dV$

In other words, the summation of the work done on the system by the set of external forces is equal to the work stored as strain energy in the elements that make up the system.

The virtual internal work in the right-hand-side of the above equation may be found by summing the virtual work done on the individual elements. The latter requires that force-displacement functions be used that describe the response for each individual element. Hence, the displacement of the structure is described by the response of individual (discrete) elements collectively. The equations are written only for the small domain of individual elements of the structure rather than a single equation that describes the response of the system as a whole (a continuum). The latter would result in an intractable problem, hence the utility of the finite element method. As shown in the subsequent sections, Equation leads to the following governing equilibrium equation for the system:

$$R = Kr + R^o$$

where

R = vector of nodal forces, representing external forces applied to the system's nodes.

K = system stiffness matrix, which is the collective effect of the individual elements' stiffness matrices: k^e.

r = vector of the system's nodal displacements.

R^o = vector of equivalent nodal forces, representing all external effects other than the nodal forces which are already included in the preceding nodal force vector R. These external effects may include distributed or concentrated surface forces, body forces, thermal effects, initial stresses and strains.

Once the supports' constraints are accounted for, the nodal displacements are found by solving the system of linear equations, symbolically:

$$r = K^{-1} (R - R^o)$$

Subsequently, the strains and stresses in individual elements may be found as follows:

$$\epsilon = Bq$$

$$\sigma = E(\epsilon - \epsilon^o) + \sigma^o = E(Bq - \epsilon^o) + \sigma^o$$

where

q = vector of a nodal displacements--a subset of the system displacement vector r that pertains to the elements under consideration.

B = strain-displacement matrix that transforms nodal displacements q to strains at any point in the element.

E = elasticity matrix that transforms effective strains to stresses at any point in the element.

ϵ^o = vector of initial strains in the elements.

σ^o = vector of initial stresses in the elements.

By applying the virtual work equation to the system, we can establish the element matrices B, k^e as well as the technique of assembling the system matrices R^o and K. Other matrices such as ϵ^o, σ^o, R and E are known values and can be directly set up from data input.

Rotational Symmetry with Respect to any Angle

Rotational symmetry with respect to any angle is, in two dimensions, circular symmetry. The fundamental domain is a half-line.

In three dimensions we can distinguish cylindrical symmetry and spherical symmetry (no change when rotating about one axis, or for any rotation). That is, no dependence on the angle using cylindrical coordinates and no dependence on either angle using spherical coordinates. The fundamental domain is a half-plane through the axis, and a radial half-line, respectively. Axisymmetric or axisymmetrical are adjectives which refer to an object having cylindrical symmetry, or axisymmetry (i.e. rotational symmetry with respect to a central axis) like a doughnut (torus). An example of approximate spherical symmetry is the Earth (with respect to density and other physical and chemical properties).

In 4D, continuous or discrete rotational symmetry about a plane corresponds to corresponding 2D rotational symmetry in every perpendicular plane, about the point of intersection. An object can also have rotational symmetry about two perpendicular planes, e.g. if it is the Cartesian product

of two rotationally symmetry 2D figures, as in the case of e.g. the duocylinder and various regular duoprisms.

Axisymmetric Boundary Conditions

In this boundary condition, the model is axisymmetric with respect to the main axis such that at a particular r = R, all θs and each z = Z-slice, each flow variable has the same value.[3] A good example is the flow in a circular pipe where the flow and pipe axes coincide.

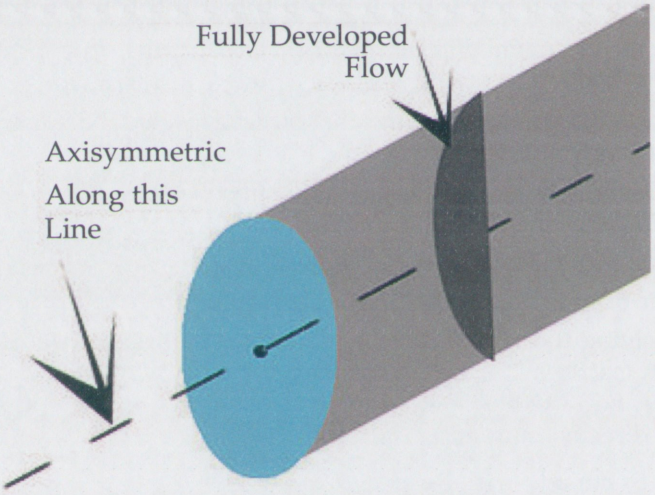


Fig. 8.1 Showing axisymmetric Boundary Condition

$$V_r (R, \theta, Z) = \text{Constant } (r = R, \theta, Z)$$

For plates and geogrids the stiffness properties must be specified in a stiffness per unit width. A common mistake is to assume that for axisymmetric problems the stiffness must be specified per radian, but this is not correct. For example, if a horizontal plate is defined on ground level one radian represents a pie slice that becomes wider when further away from the axis of symmetry. If the stiffness would be defined per radian this would mean that the stiffness of the pie slice remains constant when going further away from the axis of symmetry. However, further away from the axis of symmetry the pie slice is wider and thus has a lot more plate material. The only way to keep the stiffness constant when the plate becomes wider is either decrease the Young's modulus of the plate or decrease the thickness of the plate. Both cases are not practical: a circular plate that has been assigned one material set is expected to be of one and the same material and have constant thickness. This means the stiffness must be defined dependent on the width of the pie slice, that is per meter out-of plane.

For node-to-node anchors it is different since the anchors are not continuous in out-of-plane direction but line elements placed likes spokes in a wheel. The distance between the node-to-node anchors indeed increases when moving further away from the axis of symmetry, and according to the analogy of the wheel the number of node-to-node anchors (spokes) is constant per slice of the wheel. This means that anchor properties are per radian, and not per meter out of plane. In Plaxis the input for node to node anchors is split in two parts; a stiffness per anchor and a so called spacing that indicates the distance out of plane in between the anchors. For an axisymmetric case this spacing for node to node anchors is in radians. Note that the spacing for fixed end anchors is still per meter out of plane in an axisymmetric model.

3. HIGHER-ORDER ELEMENTS

Higher order elements can be created easily by adding additional intermediate nodes to each element edge. For an interior edge, the position of an intermediate node is determined directly by interpolation using the positions of the nodes at each end of the edge. For a boundary edge, however, the parametric position of the intermediate node between nodes n_k and n_{k+1} may be computed either by or by interpolating the parametric position of n_k and n_{k+1} directly, so that the position of the intermediate node can be mapped onto the curvilinear boundary.

These nodes are generated at the boundary node generation stage and placed on the boundary curve after the completion of the element generation process. The position of any interior nodes can be interpolated by the position of the element perimeter nodes. Figure (below) shows the locations of vertex nodes, edge nodes, and interior nodes for some quadratic elements.

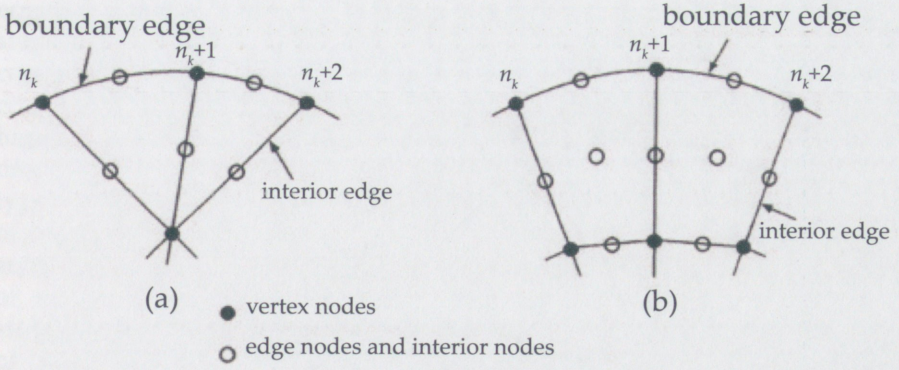


Fig. 8.2 Quadratic elements

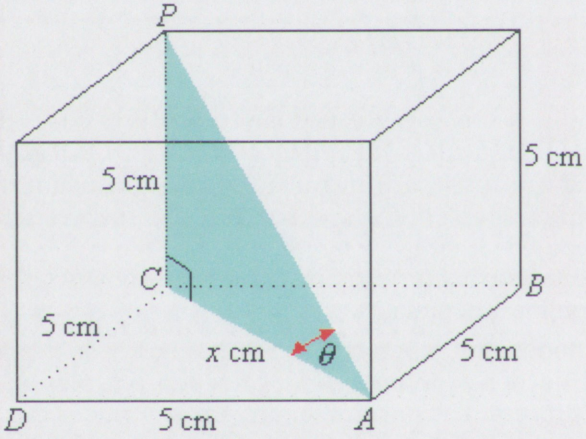
4. THREE-DIMENSIONAL PROBLEMS

To solve a three-dimensional problem, it is important to be able to visualise right triangles contained in a diagram. Then redraw the right triangles in two dimensions and use an appropriate trigonometric ratio and/or apply Pythagoras' Theorem to obtain the answer.

Find the angle between the body diagonal and the base of a cube of side-length 5 cm.

Solution:

Let θ be the angel between the diagonal AP and the base ABCD

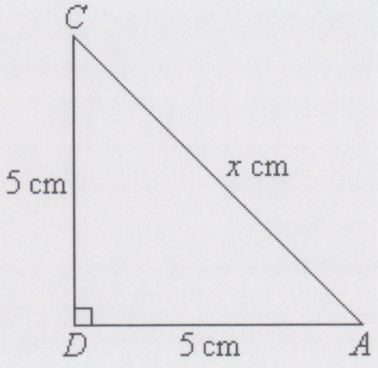


By Pythagoras' Theorem and triangle ACD,

$x^2 = 5^2 + 5^2$

$= 25 + 25$

$= 50$

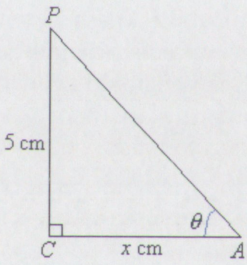


Taking the square root of both sides gives

$$x = \pm\sqrt{50}$$
$$= 7.0711 \text{ as } x > 0$$

From ΔACP, we have

$$\tan\theta = \frac{5}{x} \qquad \text{(TOA)}$$
$$= \frac{5}{7.0711}$$
$$= 0.7071$$
$$\therefore \theta = 35°16'$$



So, the angle between the body diagonal and the base of the cube is 35º16/.

5. BIHARMONIC PROBLEMS INVOLVING INCOMPRESSIBLE MATERIALS

The study of the stress-strain state of plates and shells subjected to various types of loads with different support conditions entails the formulation of boundary-value problems that generally involve systems of partial differential equations with variable coefficients. The complexity of solving these problems stems not only from the high order of the system and the variability of the coefficients, but also from the need to exactly satisfy prescribed boundary conditions. The use of a given method to obtains a solution with a satisfactorily high degree of accuracy depends to a significant extent on the geometric and mechanical parameters characterizing certain aspects of the problem and the type of boundary conditions. These factors sometimes limit the possibilities of solving problems in the important (in a practical sense) cases in which the stiffness of the shell or plate supports is also variable. In addition, problems of the shell theory entail local and edge effects, which imposes certain stiffness conditions on boundary-value problems related to the phenomenon of instability in the computation. Spline functions have recently come into wide use to solve such problems in the areas of computational mathematics, mathematical physics, and mechanics.

The popularity of this approach stems from the advantages offered by spline functions compared to other methods. Among these advantages: the stability of splines in relation to local perturbations, i.e. the behavior of the spline in the neighborhood of a point has no effect on the behavior of the spline as a whole (which is not the case in a polynomial approximation); good convergence of a spline-interpolation, in contrast to a polynomial interpolation; simplicity and ease of use of algorithms that construct and calculate splines on computers.

6. PLATE AND SHELL PROBLEMS

In continuum mechanics, plate theories are mathematical descriptions of the mechanics of flat plates that draws on the theory of beams. Plates are defined as plane structural elements with a small thickness compared to the planar dimensions.[1] The typical thickness to width ratio of a plate structure is less than 0.1. A plate theory takes advantage of this disparity in length scale to reduce the full three-dimensional solid mechanics problem to a two-dimensional problem. The aim of plate theory is to calculate the deformation and stresses in a plate subjected to loads.

Of the numerous plate theories that have been developed since the late 19th century, two are widely accepted and used in engineering. These are

- The Kirchhoff–Love theory of plates (classical plate theory)
- The Uflyand-Mindlin theory of plates (first-order shear plate theory)

Kirchhoff–Love Theory for Thin Plates

The Kirchhoff–Love theory is an extension of Euler–Bernoulli beam theory to thin plates. The theory was developed in 1888 by Love using assumptions proposed by Kirchhoff. It is assumed that a mid-surface plane can be used to represent the three-dimensional plate in two-dimensional form.

The following kinematic assumptions that are made in this theory:

- Straight lines normal to the mid-surface remain straight after deformation
- Straight lines normal to the mid-surface remain normal to the mid-surface after deformation
- The thickness of the plate does not change during a deformation.

Displacement Field

The Kirchhoff hypothesis implies that the displacement field has the form

$$u_\alpha(\mathbf{x}) = u_\alpha^0(x_1, x_2) - x_3 \frac{\partial w^0}{\partial x_\alpha} = u_\alpha^0 - x_3 w_{,\alpha}^0 \; ; \; \alpha = 1, 2$$

$$u_3(\mathbf{x}) = w^0(x_1, x_2)$$

where x_1 and x_2 are the Cartesian coordinates on the mid-surface of the undeformed plate, x_3 is the coordinate for the thickness direction, u_1^0, u_2^0 are the in-plane displacements of the mid-surface, and w_0 is the displacement of the mid-surface in the x_3 direction.

If φ_α are the angles of rotation of the normal to the mid-surface, then in the Kirchhoff–Love theory $\varphi_\alpha = w_\alpha^0$.

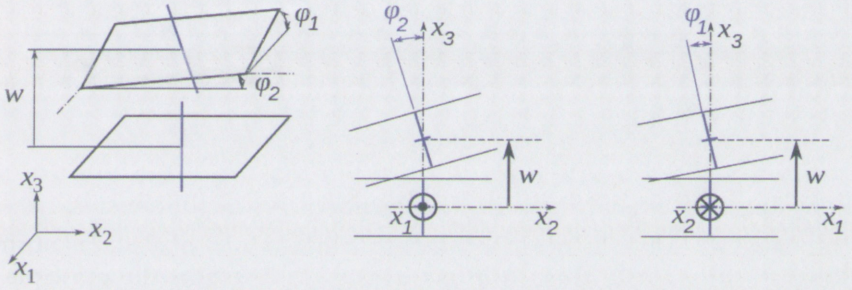


Fig. 8.3 Displacement of the mid-surface (left) and of a normal (right)

Strain-Displacement Relations

For the situation where the strains in the plate are infinitesimal and the rotations of the mid-surface normals are less than 10° the strains-displacement relations are

$$\begin{aligned}\varepsilon_{\alpha\beta} &= \tfrac{1}{2}(u^0_{\alpha,\beta} + u^0_{\beta,\alpha}) - x_3\, w^0_{,\alpha\beta} \\ \varepsilon_{\alpha 3} &= -w^0_{,\alpha} + w^0_{,\alpha} = 0 \\ \varepsilon_{33} &= 0\end{aligned}$$

Therefore, the only non-zero strains are in the in-plane directions.

If the rotations of the normals to the mid-surface are in the range of 10° to 15°, the strain-displacement relations can be approximated using the von Kármán strains. Then the kinematic assumptions of Kirchhoff-Love theory lead to the following strain-displacement relations

$$\begin{aligned}\varepsilon_{\alpha\beta} &= \frac{1}{2}(u^0_{\alpha,\beta} + u^0_{\beta,\alpha} + w^0_{,\alpha}\, w^0_{,\beta}) - x_3\, w^0_{,\alpha\beta} \\ \varepsilon_{\alpha 3} &= -w^0_{,\alpha} + w^0_{,\alpha} = 0 \\ \varepsilon_{33} &= 0\end{aligned}$$

This theory is nonlinear because of the quadratic terms in the strain-displacement relations.

Equilibrium Equations

The equilibrium equations for the plate can be derived from the principle of virtual work. For the situation where the strains and rotations of the plate are small, the equilibrium equations for an unloaded plate are given by

$$\begin{aligned} N_{\alpha\beta,\alpha} &= 0 \\ M_{\alpha\beta,\alpha\beta} &= 0 \end{aligned}$$

where the stress resultants and stress moment resultants are defined as

$$N_{\alpha\beta} := \int_{-h}^{h} \sigma_{\alpha\beta}\, dx_3 \; ; \; M_{\alpha\beta} := \int_{-h}^{h} x_3\, \sigma_{\alpha\beta}\, dx_3$$

and the thickness of the plate is 2h. The quantities $\sigma_{\alpha\beta}$ are the stresses.

If the plate is loaded by an external distributed load q(x) that is normal to the mid-surface and directed in the positive x_3 direction, the principle of virtual work then leads to the equilibrium equations

$$\begin{aligned} N_{\alpha\beta,\alpha} &= 0 \\ M_{\alpha\beta,\alpha\beta} - q &= 0 \end{aligned}$$

For moderate rotations, the strain-displacement relations take the von Karman form and the equilibrium equations can be expressed as

$$\begin{aligned} N_{\alpha\beta,\alpha} &= 0 \\ M_{\alpha\beta,\alpha\beta} + [N_{\alpha\beta}\, w^0_{,\beta}]_{,\alpha} - q &= 0 \end{aligned}$$

Boundary Conditions

The boundary conditions that are needed to solve the equilibrium equations of plate theory can be obtained from the boundary terms in the principle of virtual work.

For small strains and small rotations, the boundary conditions are

$$\begin{aligned} n_\alpha\, N_{\alpha\beta} \quad &\text{or} \quad u^0_\beta \\ n_\alpha\, M_{\alpha\beta,\beta} \quad &\text{or} \quad w^0 \\ n_\beta\, M_{\alpha\beta} \quad &\text{or} \quad w^0_{,\alpha} \end{aligned}$$

Note that the quantity $n_\alpha\, M_{\alpha\beta,\beta}$ is an effective shear force.

Stress–Strain Relations

The stress–strain relations for a linear elastic Kirchhoff plate are given by

$$\begin{bmatrix} \sigma_{11} \\ \sigma_{22} \\ \sigma_{12} \end{bmatrix} = \begin{bmatrix} C_{11} & C_{12} & C_{13} \\ C_{12} & C_{22} & C_{23} \\ C_{13} & C_{23} & C_{33} \end{bmatrix} \begin{bmatrix} \varepsilon_{11} \\ \varepsilon_{22} \\ \varepsilon_{12} \end{bmatrix}$$

Since $\sigma_{\alpha 3}$ and σ_{33} do not appear in the equilibrium equations it is implicitly assumed that these quantities do not have any effect on the momentum balance and are neglected.

It is more convenient to work with the stress and moment resultants that enter the equilibrium equations. These are related to the displacements by

$$\begin{bmatrix} N_{11} \\ N_{22} \\ N_{12} \end{bmatrix} = \left\{ \int_{-h}^{h} \begin{bmatrix} C_{11} & C_{12} & C_{13} \\ C_{12} & C_{22} & C_{23} \\ C_{13} & C_{23} & C_{33} \end{bmatrix} dx_3 \right\} \begin{bmatrix} u^0_{1,1} \\ u^0_{2,2} \\ \frac{1}{2}(u^0_{1,2} + u^0_{2,1}) \end{bmatrix}$$

and

$$\begin{bmatrix} M_{11} \\ M_{22} \\ M_{12} \end{bmatrix} = -\left\{ \int_{-h}^{h} x_3^2 \begin{bmatrix} C_{11} & C_{12} & C_{13} \\ C_{12} & C_{22} & C_{23} \\ C_{13} & C_{23} & C_{33} \end{bmatrix} dx_3 \right\} \begin{bmatrix} w^0_{,11} \\ w^0_{,22} \\ w^0_{,12} \end{bmatrix}.$$

The extensional stiffnesses are the quantities

$$A_{\alpha\beta} := \int_{-h}^{h} C_{\alpha\beta}\, dx_3$$

The bending stiffnesses (also called flexural rigidity) are the quantities

$$D_{\alpha\beta} := \int_{-h}^{h} x_3^2\, C_{\alpha\beta}\, dx_3$$

7. ISOPARAMETRIC ELEMENTS

Isoparametric elements are formulations for finite elements used in the finite element method (FEM), which are widely used in the analysis of solids. The FEM is a numerical method for the solution of differential equations

whose unique solution requires certain additional conditions, so-called initial and boundary conditions . The solution of the initial boundary value problem is developed in the FEM in approach functions that are potentially able to approximate every solution function allowed for comparison. A discretization error occurs due to the limitation of the shape functions to a finite number.

Now, the solution is split additively into two parts: one that reflects the initial condition and the other that has to be adapted to the boundary conditions. Now one could approximate these two parts with different shape functions. The special thing about the isoparametric elements is that they use the same approach functions for both parts of the solution. The standard formulation of the FEM is the displacement method. In order to be able to represent a location-independent displacement field there, the approach functions must meet certain requirements that are particularly easy to meet in isoparametric elements. Isoparametric elements have the advantages that

1. They can be used in most problems,
2. A constant displacement field can be easily represented in them,
3. The discretization error can be easily assessed visually in the preprocessor and
4. only one set of approach functions needs to be defined and programmed.

Solution of Differential Equations

The FEM is a method for the solution of differential equations (Dgl) and therefore the solution of such equations should be sketched here with a small example. The equation of motion of a mass point with mass m at which a force F acts is according to Newton's second law

$$m\ddot{\chi}(t) - F = 0 .$$

The location of the mass point is given by the motion function χ (t) as a function of time t and the two attached points symbolize the second time derivative (acceleration). We are looking for a solution in the time interval. With constant force, this Dgl can be integrated closed:

$$\ddot{\chi}(t) = \frac{F}{m} \rightarrow \dot{\chi}(t) = v_0 + \frac{F}{m}t \rightarrow \chi(t) = X + v_0 t + \frac{F}{2m}t^2 .$$

In addition to the function $\chi(t)$, the mass m and the time t, this solution contains three variables: X, v_0 and F, which have to be specified in a concrete calculation case. The conditions prevailing at the start of the movement are the initial conditions , which are determined here by the position X and speed v_0 at time t = 0. The force F is one of the boundary conditions that are generally dependent on time and place.

The functions $\chi_h(t)$, which at time t = 0, the homogeneous initial and boundary conditions

$$\chi_h(0)=0 \quad \wedge \quad \dot{\chi}_h(0)=0$$

suffice, can be added in any way to a given solution of the Dgl without violating these initial and boundary conditions. In this example, the force would be constant

$$\chi_h(t)=\frac{t^2}{2} \rightarrow \chi_h(0)=\dot{\chi}_h(0)=0, \quad \ddot{\chi}_h(t)=1.$$

The particular solution $\chi_p(t)$, however, satisfies the inhomogeneous initial and boundary conditions

$$\chi_p(0)=X \quad \wedge \quad \dot{\chi}_p(0)=v_0 \quad \rightarrow \quad \chi_p(t)=X+v_0 t.$$

The complete solution of the Dgl is the sum of the particulate solution and the homogeneous proportions of the solution:

$$\chi(t)=\chi_p(t)+\frac{F}{m}\chi_h(t)=X+v_0 t+\frac{F}{2m}t^2.$$

The displacement of the body is the difference between its current position χ (t) and its starting position X and is therefore the time-dependent part of the solution function

$$u(t)=\chi(t)-X=v_0 t+\frac{F}{2m}t^2,$$

whereas the starting position X = χ (t) -u (t) is the time-independent part of the solution function. In the approximate solution of a Dgl using the method of weighted residuals (Ritz's variation method), a functional approach is made for the solution sought, with which the exact solution can be approximated, e.g. B .:

$$\chi^h(t)=At^2 \quad \rightarrow \quad \dot{\chi}^h(t)=2At \rightarrow \ddot{\chi}^h(t)=2A.$$

This approach is used in the Dgl, which is usually not met exactly. The rest is the residual

$$R(t)=m\ddot{\chi}^h(t)-F=2mA-F.$$

The factor A is calculated according to the Galerkin approach by multiplying the residue by the approach function, integrating it over the interval of interest, here [0.1], and making this weighted remainder disappear:

$$\int_0^1 t^2(2mA-F)dt=(2mA-F)\left[\frac{t^3}{3}\right]_0^1=\frac{1}{3}(2mA-F)\overset{!}{=}0 \rightarrow A=\frac{F}{2m},$$

This creates the homogeneous approximate solution

$$\chi_h(t) = \frac{F}{2m} t^2$$

This has to be supplemented by the particulate components in order to take into account the initial and boundary conditions:

$$\chi(t) = \chi_p(t) + \frac{F}{2m} t^2 = X + v_0 t + \frac{F}{2m} t^2.$$

The Finite Element Method

In the classic Ritz approach, the solution function is replaced by a sum of functions that depend on a parameter vector and can approximate every function allowed for comparison. The solution to the Dgl is thus reduced from the search for a function to the search for a parameter vector.

The initial positions of the particles - the geometry of the body - is, as stated, a particulate solution component of the Dgl that obeys the initial conditions. Simplifications result from the following assumptions of the FEM that go beyond the Ritz method and the Galerkin approach:

1. Since finding the particular solution is difficult in most cases, this, like the solution function, is also developed in sums of functions. The parameters of the approach functions are to be chosen so that the initial and boundary conditions (body geometry, initial speed) are satisfied.
2. The time and location dependency of the batch functions is divided into purely location-dependent functions and only time-dependent parameters.

In the engineer's approach, physically significant quantities such as the coordinates or displacements of certain points are selected as parameters. This makes it easy to determine the parameters and to check their plausibility.

Standard Formulation

The first application of FEM was the linear treatment of solids and structures and it was based on this that FEM received its impulses. When computing problems in solid mechanics , displacements are sought that reflect the translation, rotation and deformation of a body. During the development of the FEM, a large number of possibilities for describing the geometry and the displacements were shown. The standard formulation is the displacement method in which the displacements at selected points, the nodes, are selected as parameters and the displacements between the nodes are interpolated with shape functions.

The so-called mixed methods use not only the knot displacements but also other unknowns that physically correspond to a pressure, tension or strain. The element shifts then result from the interaction of all these variables. The isoparametric elements establish the connection between the element displacements and the node displacements directly. In addition to the advantages mentioned above, the isoparametric elements also have the additional advantage that the displacement field can be easily visualized.

8. NONLINEAR PROBLEMS

Nonlinear problems are of interest to engineers, biologists, physicists, mathematicians, and many other scientists because most systems are inherently nonlinear in nature. Nonlinear dynamical systems, describing changes in variables over time, may appear chaotic, unpredictable, or counterintuitive, contrasting with much simpler linear systems.

Typically, the behavior of a nonlinear system is described in mathematics by a nonlinear system of equations, which is a set of simultaneous equations in which the unknowns (or the unknown functions in the case of differential equations) appear as variables of a polynomial of degree higher than one or in the argument of a function which is not a polynomial of degree one. In other words, in a nonlinear system of equations, the equation(s) to be solved cannot be written as a linear combination of the unknown variables or functions that appear in them. Systems can be defined as nonlinear, regardless of whether known linear functions appear in the equations. In particular, a differential equation is linear if it is linear in terms of the unknown function and its derivatives, even if nonlinear in terms of the other variables appearing in it.

As nonlinear dynamical equations are difficult to solve, nonlinear systems are commonly approximated by linear equations (linearization). This works well up to some accuracy and some range for the input values, but some interesting phenomena such as solitons, chaos, and singularities are hidden by linearization. It follows that some aspects of the dynamic behavior of a nonlinear system can appear to be counterintuitive, unpredictable or even chaotic. Although such chaotic behavior may resemble random behavior, it is in fact not random. For example, some aspects of the weather are seen to be chaotic, where simple changes in one part of the system produce complex effects throughout. This nonlinearity is one of the reasons why accurate long-term forecasts are impossible with current technology.

Types of Nonlinear Dynamic Behaviors

- **Amplitude death** – any oscillations present in the system cease due to some kind of interaction with other system or feedback by the same system

- **Chaos** – values of a system cannot be predicted indefinitely far into the future, and fluctuations are aperiodic
- **Multistability** – the presence of two or more stable states
- **Solitons** – self-reinforcing solitary waves
- **Limit cycles** – asymptotic periodic orbits to which destabilized fixed points are attracted.
- **Self-oscillations** - feedback oscillations taking place in open dissipative physical systems.

Nonlinear Algebraic Equations

Nonlinear algebraic equations, which are also called polynomial equations, are defined by equating polynomials (of degree greater than one) to zero. For example,

$$x^2 + x - 1 = 0.$$

For a single polynomial equation, root-finding algorithms can be used to find solutions to the equation (i.e., sets of values for the variables that satisfy the equation). However, systems of algebraic equations are more complicated; their study is one motivation for the field of algebraic geometry, a difficult branch of modern mathematics. It is even difficult to decide whether a given algebraic system has complex solutions (see Hilbert's Nullstellensatz). Nevertheless, in the case of the systems with a finite number of complex solutions, these systems of polynomial equations are now well understood and efficient methods exist for solving them.

Nonlinear Recurrence Relations

A nonlinear recurrence relation defines successive terms of a sequence as a nonlinear function of preceding terms. Examples of nonlinear recurrence relations are the logistic map and the relations that define the various Hofstadter sequences. Nonlinear discrete models that represent a wide class of nonlinear recurrence relationships include the NARMAX (Nonlinear Autoregressive Moving Average with eXogenous inputs) model and the related nonlinear system identification and analysis procedures. These approaches can be used to study a wide class of complex nonlinear behaviors in the time, frequency, and spatio-temporal domains.

Nonlinear Differential Equations

A system of differential equations is said to be nonlinear if it is not a linear system. Problems involving nonlinear differential equations are extremely diverse, and methods of solution or analysis are problem dependent. Examples of nonlinear differential equations are the Navier–Stokes equations in fluid dynamics and the Lotka–Volterra equations in biology.

One of the greatest difficulties of nonlinear problems is that it is not generally possible to combine known solutions into new solutions. In linear problems, for example, a family of linearly independent solutions can be used to construct general solutions through the superposition principle. A good example of this is one-dimensional heat transport with Dirichlet boundary conditions, the solution of which can be written as a time-dependent linear combination of sinusoids of differing frequencies; this makes solutions very flexible. It is often possible to find several very specific solutions to nonlinear equations, however the lack of a superposition principle prevents the construction of new solutions.

Ordinary Differential Equations

First order ordinary differential equations are often exactly solvable by separation of variables, especially for autonomous equations. For example, the nonlinear equation

$$\frac{du}{dx} = -u^2$$

has $u = \frac{1}{x + C}$ as a general solution (and also $u = 0$ as a particular solution, corresponding to the limit of the general solution when C tends to infinity). The equation is nonlinear because it may be written as

$$\frac{du}{dx} + u^2 = 0$$

and the left-hand side of the equation is not a linear function of u and its derivatives. Note that if the u^2 term were replaced with u, the problem would be linear (the exponential decay problem).

Second and higher order ordinary differential equations (more generally, systems of nonlinear equations) rarely yield closed-form solutions, though implicit solutions and solutions involving nonelementary integrals are encountered.

Common methods for the qualitative analysis of nonlinear ordinary differential equations include:

- Examination of any conserved quantities, especially in Hamiltonian systems
- Examination of dissipative quantities (see Lyapunov function) analogous to conserved quantities
- Linearization via Taylor expansion
- Change of variables into something easier to study
- Bifurcation theory
- Perturbation methods (can be applied to algebraic equations too)

Partial Differential Equations

The most common basic approach to studying nonlinear partial differential equations is to change the variables (or otherwise transform the problem) so that the resulting problem is simpler (possibly even linear). Sometimes, the equation may be transformed into one or more ordinary differential equations, as seen in separation of variables, which is always useful whether or not the resulting ordinary differential equation(s) is solvable.

Another common (though less mathematical) tactic, often seen in fluid and heat mechanics, is to use scale analysis to simplify a general, natural equation in a certain specific boundary value problem. For example, the (very) nonlinear Navier-Stokes equations can be simplified into one linear partial differential equation in the case of transient, laminar, one dimensional flow in a circular pipe; the scale analysis provides conditions under which the flow is laminar and one dimensional and also yields the simplified equation.

Other methods include examining the characteristics and using the methods outlined above for ordinary differential equations.

Pendula

A classic, extensively studied nonlinear problem is the dynamics of a pendulum under the influence of gravity. Using Lagrangian mechanics, it may be shown[13] that the motion of a pendulum can be described by the dimensionless nonlinear equation

$$\frac{d^2\theta}{dt^2} + \sin(\theta) = 0$$

where gravity points "downwards" and θ is the angle the pendulum forms with its rest position, as shown in the figure at right. One approach to "solving" this equation is to use $d\theta/dt$ as an integrating factor, which would eventually yield

$$\int \frac{d\theta}{\sqrt{C_0 + 2\cos(\theta)}} = t + C_1$$

which is an implicit solution involving an elliptic integral. This "solution" generally does not have many uses because most of the nature of the solution is hidden in the nonelementary integral (nonelementary unless $C_0 = 2$.

Another way to approach the problem is to linearize any nonlinearities (the sine function term in this case) at the various points of interest through Taylor expansions. For example, the linearization at $\theta = 0$, called the small angle approximation, is

$$\frac{d^2\theta}{dt^2} + \theta = 0$$

since $\sin(\theta) \approx \theta$ for $\theta \approx 0$. This is a simple harmonic oscillator corresponding to oscillations of the pendulum near the bottom of its path. Another linearization would be at $\theta = \pi$, corresponding to the pendulum being straight up:

$$\frac{d^2\theta}{dt^2} + \pi - \theta = 0$$

since $\sin(\theta) \approx \pi - \theta$ for $\theta \approx \pi$. The solution to this problem involves hyperbolic sinusoids, and note that unlike the small angle approximation, this approximation is unstable, meaning that $|\theta|$ will usually grow without limit, though bounded solutions are possible. This corresponds to the difficulty of balancing a pendulum upright, it is literally an unstable state.

One more interesting linearization is possible around $\theta = \pi/2$, around which $\sin(\theta) \approx 1$:

$$\frac{d^2\theta}{dt^2} + 1 = 0.$$

This corresponds to a free fall problem. A very useful qualitative picture of the pendulum's dynamics may be obtained by piecing together such linearizations, as seen in the figure at right. Other techniques may be used to find (exact) phase portraits and approximate periods.

CONCLUSION

The finite element solutions to some fairly simple problems in electrostatics have been presented in a step-by-step fashion. Hopefully, these examples will allow you to apply the program to practical problems with more complicated geometries. Practical optimization problems frequently involve nonlinear behavior that must be taken into account. It is sometimes possible to reformulate these nonlinearities to fit into a linear programming format, as can be done for separable programming problems. However, it is frequently necessary to use a nonlinear programming formulation.

A method for solving a system of nonlinear equations is proposed, which is converted to an optimization problem, and an improved particle swarm algorithm is employed to solve the optimization problem. This method overcomes the dependence on reasonable initial guess of the solution. And then some standard systems of nonlinear equations are presented to demonstrate the convergence probability, convergence rate, and solution precision in finding the best solution of nonlinear equations with the improved particle swarm algorithm.

REFERENCES

- Andrews, L.C. (1985) Special Functions for Engineers and Applied Mathematicians, Macmillan, New York.
- Boyce, W.E. and DiPrima, R.C. (1992) Elementary Differential Equations and Boundary Value Problems, 5th edition, John Wiley, New York.
- Brown, J.W. and Churchill, R.V. (1993) Fourier Series and Boundary Value Problems, McGraw-Hill, New York.
- Byerly, W.E. (1959) Fourier Series, Dover Publications, New York.
- Carslaw, H.S. (1980) An Introduction to the Theory of Fourier Series and Integrals, Dover Publications, New York.
- Churchill, R.V. (1960) Complex Variables and Applications, McGraw-Hill, New York.
- Davis, H.F. (1963) Fourier Series and Orthogonal Functions, Allyn and Bacon, Boston.
- Doetsch, G. (1963) Guide to the Applications of the Laplace Transform, Van Nostrand, New York.
- Edwards, R.E. (1967) Fourier Series: A Modern Introduction, Holt, Rinehart and Winston, New York.
- Franklin, P. (1949) An Introduction to Fourier Methods and the Laplace Transformation, Dover Publications, New York.
- Hochstadt, H. (1961) Special Functions of Mathematical Physics, Holt Rinehart, New York.
- Potter, M.C. (1978) Mathematical Methods in the Physical Sciences, Prentice-Hall, Englewood Cliffs, N.J.
- Riley, K.F., Hobson, M.P. and Bence, S.J. (1998) Mathematical Methods for Physics and Engineering, Cambridge University Press, Cambridge.
- Spencer, A.J.M., et al. (1977) Engineering Mathematics, Vols. 1 and 2, Van Nostrand Reinhold, New York.
- Spiegel, M.R. (1971) Advanced Mathematics for Engineers and Scientists, McGraw-Hill, New York.
- Strang, G. (1986) Introduction to Applied Mathematics, Wellesley-Cambridge Press, Cambridge, Mass. von Karman, T. and Biot, M.A. (1940) Mathematical Methods in Engineering, McGrawHill, New York.
- Wylie, C. (1958) Advanced Engineering Mathematics, McGraw-Hill, New York. Fourier Series, Integral Transforms, Special Functions and Complex Variables

MCQs

1. **Axisymmetry implies that points lying on the z- axis remains _____ fixed.**

   a. Tangentially
   b. Spherically
   c. Radially
   d. Circularly

2. **Modeling of a cylinder of infinite length subjected to external pressure. The length dimensions are assumed to be _____**
   a. Finite
   b. Non uniform
   c. Perpendicular
   d. Constant
3. **Press fit of a ring of length L and internal radius r_j onto a rigid shaft of radius $r_1+\delta$ is considered. When symmetry is assumed about the mid plane, this plane is restrained in the _____**
   a. X direction
   b. Y direction
   c. Z direction
   d. Undefined
4. **The condition that nodes at the internal radius have to displace radially by δ , a large stiffness C is added to the _____**
   a. Co-ordinates
   b. Length
   c. Diagonal locations
   d. Radius
5. **Press fit on elastic shaft, may define pairs of nodes on the contacting boundary, each pair consisting of one node on the _____ and one on the _____**
   a. Shaft and couple
   b. Sleeve and shaft
   c. Shaft and sleeve
   d. Sleeve and couple
6. **Consider an Axisymmetric problem of which is having a radius of r, rotational angle θ and length l. Then r dl dθ is known as ____**
   a. Elemental volume
   b. Element
   c. Elemental surface area
   d. Elemental surface
7. **The stress vector is correspondingly defined as _____**
   a. $\sigma=[\sigma_y,\sigma_z,\tau_{yz},\sigma_\theta]^T$
   b. $\sigma=[\sigma_y,\sigma_z]^T$

c. $u=[u, w]^T$

d. $T=[T_y T_z]^T$

8. **Problems involving three- dimensional axisymmetric solids or solids of revolution, subjected to _____ loading.**

a. Rotational

b. Two-dimensional

c. Three-dimensional

d. Axisymmetric loading

9. **All deformations and stresses are independent of _____**

a. Co-ordinates

b. Number of nodes

c. Rotational angle, θ

d. Area

10. **Revolving bodies like fly wheels can be analyzed by introducing _____ in body force term.**

a. Gravitational force

b. Revolving force

c. Centrifugal force

d. Centripetal force

11. **The stress- strain relation is given as _____**

a. $\sigma=D$

b. $\sigma=D\varepsilon$

c. $\sigma=\varepsilon$

d. $\varepsilon=D\sigma$

12. **Axisymmetric problems are totally defined in _____**

a. xy planes

b. yz planes

c. rz planes

d. $r\theta$ planes

ANSWERS

1 (c)	2 (d)	3 (c)	4 (c)	5 (b)	6 (c)
7 (a)	8 (d)	9 (c)	10 (c)	11 (b)	12 (c)

9 FEA of Elliptic Boundary Value Problems

OBJECTIVES

After studying this chapter, you will be able to know:

- Galerkin Approximation
- Existence and Uniqueness of Galerkin Approximations
- Finite-Element Approximations
- Properties of Finite-Element Subspaces
- Error Estimates
- Quadrature, Boundary and Data Errors

1. INTRODUCTION

A finite element numerical method is investigated for fractional order elliptic boundary value problems with homogeneous Dirichlet type boundary conditions. It is pointed out that an appropriate stiffness matrix can be obtained by taking the prescribed fractional power of the stiffness matrix corresponding to the non-fractional elliptic operators. It is proved that this approach, which is also called the matrix transformation or matrix transfer method, delivers optimal rate of convergence in the L_2-norm.

Finite element method is an important numerical method for the problems of partial differential equations and widely used in the numerical solution of optimal control problems. There are extensive studies in convergence of finite element approximation for optimal control problems. For example, priori error estimates for finite element discretization of optimal control problems governed by elliptic equations are discussed in many publications. In a new approach to error control and mesh adaptivity is described for the discretization of the optimal control problems governed by elliptic partial differential equations.

In the error estimates for semilinear elliptic optimal controls in the maximum norm are presented. Chen and Liu present a priori error analysis for mixed finite element approximation of quadratic optimal control problems. In a priori error analysis for the finite element discretization of the optimal control problems governed by elliptic state equations is considered. Hou and Li investigate the error estimates of mixed finite element methods for optimal control problems governed by general elliptic equations and derive and error estimates for both the control and state variables.

2. GALERKIN APPROXIMATION

A Galerkin method for a modified regularized long wave equation is studied using finite elements in space, the Crank-Nicolson scheme, and the Runge-Kutta scheme in time. In addition, an extrapolation technique is used to transform a nonlinear system into a linear system in order to improve the time accuracy of this method. A Fourier stability analysis for the method is shown to be marginally stable. Three invariants of motion are investigated. Numerical experiments are presented to check the theoretical study of this method.

The approach is usually credited to Boris Galerkin. The method was explained to the Western reader by Hencky and Duncan among others. Its convergence was studied by Mikhlin and Leipholz Its coincidence with Fourier method was illustrated by Elishakoff et al. Its equivalence to Ritz's method for conservative problems was shown by Singer. Gander and Wanner showed how Ritz and Galerkin methods led to the modern finite element method. One hundred years of method's development was discussed by Repin. Often when referring to a Galerkin method, one also gives the name along with typical approximation methods used, such as Bubnov–Galerkin method (after Ivan Bubnov), Petrov–Galerkin method (after Georgii I. Petrov) or Ritz–Galerkin method (after Walther Ritz).

Analysis of Galerkin Methods

Here, we will restrict ourselves to symmetric bilinear forms, that is

$$a(u, v) = a(v, u).$$

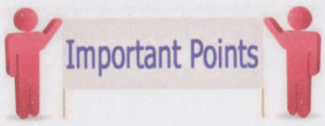
While this is not really a restriction of Galerkin methods, the application of the standard theory becomes much simpler. Furthermore, a Petrov–Galerkin method may be required in the nonsymmetric case.

The analysis of these methods proceeds in two steps. First, we will show that the Galerkin equation is a well-posed problem in the sense of Hadamard and therefore admits a unique solution. In the second step, we study the quality of approximation of the Galerkin solution u_n.

The analysis will mostly rest on two properties of the bilinear form, namely

Boundedness: for all $u, v \in V$ holds

$$a(u, v) \leq C \| u \| \| v \|$$



A Fourier stability analysis for the method is shown to be marginally stable.

Ellipticity: for all $u \in V$ holds

$$a(u,u) \geq c \| u \|^2 \text{ for some constant } c > 0.$$

By the Lax-Milgram theorem (see weak formulation), these two conditions imply well-posedness of the original problem in weak formulation. All norms in the following sections will be norms for which the above inequalities hold (these norms are often called an energy norm).

Well-Posedness of the Galerkin Equation

Since $V_n \in V$, boundedness and ellipticity of the bilinear form apply to V_n. Therefore, the well-posedness of the Galerkin problem is actually inherited from the well-posedness of the original problem.

Quasi-Best Approximation (Céa's lemma)

The error $u - u_n$ between the original and the Galerkin solution admits the estimate

$$\| u - u_n \| \leq \frac{C}{c} \inf_{v_n \in V_n} \| u - v_n \|.$$

This means, that up to the constant C/c, the Galerkin solution u_n is as close to the original solution u as any other vector in V_n. In particular, it will be sufficient to study approximation by spaces V_n, completely forgetting about the equation being solved.

Proof

Since the proof is very simple and the basic principle behind all Galerkin methods, we include it here: by ellipticity and boundedness of the bilinear form (inequalities) and Galerkin orthogonality (equals sign in the middle), we have for arbitrary $v_n \in V_n$:

$$c \| u - u_n \|^2 \leq a(u - u_n, u - u_n) = a(u - u_n, u - v_n) \leq C \| u - u_n \| \| u - v_n \|.$$

Dividing by $c \| u - u_n \|$ and taking the infimum over all possible v_n yields the lemma.

3. EXISTENCE AND UNIQUENESS OF GALERKIN APPROXIMATIONS

The initial work devoted to the second-order partial differential equations with nonlocal integral conditions goes back to Cannon. Subsequent investigations include and many other references therein. Nonlocal problems for some classes of partial differential equations have attracted much interest in the last few years, and several papers have been devoted to this subject. The problem of existence and uniqueness of solutions is of growing interest,

as well as methods of obtaining explicit or approximate solutions for this kind of problems. The main reason of such a wide interest in this type of problems is that the non-local conditions occur in various equations of mathematical physics and in mathematical biology.

Nonlocal conditions come up when values of the function on the boundary are connected to values inside the domain. There are various types of non-local integral conditions for hyperbolic, parabolic, or elliptic equations; we point out an important non-local problem, studied for a parabolic equation with the following non local conditions.

$$u(0,t) = \int_0^1 \alpha_1(x)u(x,t)dx + d_1(t),$$

$$u(0,t) = \int_0^1 \alpha_2(x)u(x,t)dx + d_2(t),$$

connected with quasistationary thermoelasticity theory. The non-local boundary conditions of type arise in modeling the technology of integral circuits, studied in.

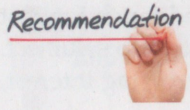
$$\alpha(t) = \int_0^1 u(x)u(x,t)dx, \quad u(1,t) = 0,$$

Problems with nonlocal integral conditions for multidimensional hyperbolic equations have been intensively studied in recent years.

Finite-Element Approximations

The computation of quadratic functionals of the solution to a linear stochastic partial differential equation with multiplicative noise is considered. An operator valued Lyapunov equation, whose solution admits a deterministic representation of the functional, is used for this purpose and error estimates are shown in suitable operator norms for a fully discrete approximation of this equation.

Weak error rates are also derived for a fully discrete approximation of the stochastic partial differential equation, using the results obtained from the approximation of the Lyapunov equation. In the setting of finite element approximations, a computational complexity comparison reveals that appro ximating the Lyapunov equation allows for cheaper computation of quadratic functionals compared to applying Monte Carlo or covariance-based methods directly to the discretized stochastic partial differential equation. Numerical simulations illustrate the theoretical results.



The problem of existence and uniqueness of solutions is of growing interest, as well as methods of obtaining explicit or approximate solutions for this kind of problems.

The rate of convergence of the finite element method depends on the order to which the solutionu can be approximated by the trial space of piecewise polynomials. We attempt to unify the many published estimates, by proving that if the trial space is complete through polynomials of degreek–1, then it contains a functionv ^{h} such that $|u-v^{h}|_{s} \in ch^{k-s}|u|_{k}$. The derivatives of orders andk are measured either in the maximum norm or in the mean-square norm, and the estimate can be made local: the error in a given element depends on the diameterh $_{i}$ of that element. The proof applies to domains Ω in any number of dimensions, and employs a uniformity assumption which avoids degenerate element shapes.

Error Estimates

In this section, it will focus on two types of error estimates for the finite element method, a priori and a posteriori estimates. (We have previously introduced these terms in the context of quadrature rules.) These two types of estimates serve very different purposes. The main feature of a priori estimates is that they tell us the order of convergence of a given finite element method, that is, they tell us that the finite element error $\|u-u_h\|$ in some norm $\|\cdot\|$ is $O(h^a)$, where h is the (maximum) mesh size and a is a positive integer. The constant in the O(his generally unknown and is often not of great interest. The goal of these estimates is to give us a reasonable measure of the efficiency of a given method by telling us how fast the error decreases as we decrease the mesh size. In contrast, a posteriori estimates use the computed solution uh in order to give us an estimate of the form $\|u-u_h\| \leq \in$, where$\in$is simply a number. These estimates accomplish two main goals. First, they are able to give us a much better idea of the actual error in a given finite element computation than are a priori estimates. Secondly, they can be used to perform adaptive mesh refinement. In adaptive mesh refinement, a posteriori error estimators are used to indicate where the error is particularly high, and more mesh intervals are then placed in those locations. A new finite element solution is computed, and the process is repeated until a satisfactory error tolerance is reached.

A Priori Error Estimates

Our goal is to find bounds for the error u − uh in the finite element approximation of the solution u to our general two-point boundary value problem. We first consider how we should measure the error, that is, what norm

IMPORTANT

An operator valued Lyapunov equation, whose solution admits a deterministic representation of the functional, is used for this purpose and error estimates are shown in suitable operator norms for a fully discrete approximation of this equation.

we should use. We have already demonstrated that the energy norm k·keng is convenient in one sense because $\|u-u_h\|_{L^\infty} \leq$ tol. eng $= \min\chi \in Sh \|u\|_e = (a(u,u))^{1/2}, \quad \forall u \in V,$.

However, sometimes we wish to measure only the error $u - u_h$ in the function values, and in fact this error converges at a higher rate than does the error in the derivative (or more precisely, the error in the H^1 or energy norms). Finally, the energy norm measures an average error in some sense; more precisely, it measures weighted integrals of the squares of the errors over the interval [0, 1]. However, controlling the average error leaves open the possibility that the error is very large on a very small portion of the interval [0, 1]. Thus we could seek to measure the maximum of the error over the interval, i.e., $\|u-u_h\|_{L^\infty} \leq$ tol. $_{L^\infty{}^1([0,1])}$.

A posteriori error estimation and adaptive mesh refinement

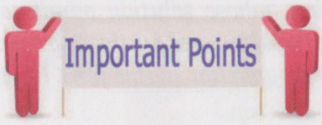
In this section, we shall try to solve the following problem: How should the mesh points $\{x_i\}_{i=0,\ldots,N}$ in a finite element mesh be placed so that a given error tolerance is achieved with maximum efficiency, i.e., with as few mesh points as possible? The answer to this problem depends on many factors, including the coefficients and solution of the given TPBVP, and in fact it is solved in different ways in different circumstances. We give here one solution which is very widely used in practice.

Thus, we shall use the following strategy. We will first compute a finite element approximation on a relatively coarse grid (i.e., a grid with relatively few mesh points). We shall use information from this computed solution to guess where more mesh points should be placed, then recompute the finite element solution on the resulting finer mesh. This procedure is continued until a given error tolerance is reached.

To be more precise, let's suppose that we are seeking to find a finite element approximation u_h to the solution u of a TPBVP such that

$$\|u-u_h\|_{L^\infty} \leq \text{tol.}$$

Here tol might be 0.01, 0.001, or something like that. It is difficult to know absolutely for certain that we have reached this goal (unless we know u ahead of time, which we generally don't), so we shall instead settle for being reasonably sure that we are close to it.



The main feature of a priori estimates is that they tell us the order of convergence of a given finite element method.

Proposition: Under the previous assumptions, we have the orthogonality identity:

$$\forall v_h \in V_h,\ a(u - u_h, v_h) = 0.$$

Proof.

Since Vh Ì V, we have trivially that $a(u, v_h) = l(v_h) = a(u_h, v_h)$, for all v_h Î Vh. If the bilinear form $a(\cdot, \cdot)$ is symmetric, continuous and V-elliptic on V ×V of constant a, then it defines an inner product and an energy norm associated to it as:

$$\|u\|_e = (a(u,u))^{1/2}, \quad \forall u \in V,$$

equivalent to the norm of V:

$$\sqrt{\alpha}\,\|u\|V \le \|u\|_e \le \sqrt{\|\alpha\|}\,\|u\|V, \quad \forall u \in V.$$

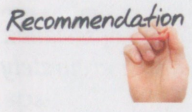
The approximate solution u_h is thus the orthogonal projection for the inner product $a(\cdot, \cdot)$, of the solution u on the subspace V_h.

A strong advantage of the internal variational approximation is that it provides an optimal estimate of the error between the exact solution u of the problem (P) and the approximate solution uh of the problem (P_h). The error $\|u - u_h\|_V$ is comparable to the minimum of $\|u - u_h\|_V$ when v_h covers V_h. This error estimate in norm $\|\cdot\|_V$ is given by C´ea's lemma that we recall here.

Quadrature, Boundary and Data Errors

In boundary integral methods it is often necessary to evaluate layer potentials on or close to the boundary, where the underlying integral is difficult to evaluate numerically. Quadrature by expansion (QBX) is a new method for dealing with such integrals, and it is based on forming a local expansion of the layer potential close to the boundary. In doing so, one introduces a new quadrature error due to nearly singular integration in the evaluation of expansion coefficients.

This makes it possible to derive accurate estimates for the quadrature errors related to QBX, when applied to layer potentials in two and three dimensions. As examples we derive estimates for the Laplace and Helmholtz single layer potentials. These results can be used for parameter selection in practical applications.



Knowing where to place mesh points before computing a finite element solution is difficult because it generally requires some knowledge about the solution–which is exactly what we don't know since that's what we're trying to compute in the first place.

A boundary quadrature formula is an approximate integration formula with all of its evaluation points lying on the boundary of the integration domain. Such a formula may be particularly useful for the case where the values of the integrand function and its derivatives inside the domain are not given or are not easily determined.

Boundary quadrature formulas are not really new. Indeed, from the viewpoint of numerical analysis, the classical Euler–Maclaurin summation formula and the Hermite two-end multiple nodes quadrature formulas may be regarded as one-dimensional boundary quadrature formulas since they make use of only the integrand function values and their derivatives at the limits of integration.

The earliest example of a boundary quadrature formula with some algebraic precision for multivariate integration is possibly the formula of algebraic precision (or degree) 5 for a triple integral over a cube given by Sadowski in 1940.

He used 42 points on the surface of a cube to construct the quadrature, which has been modified by the first author of this paper to a quadrature with 26 points, the fewest possible boundary points. Some 20 years later, Levin and Ionescu individually investigated certain optimal boundary quadrature formulas for double integration over a square using partial derivatives at some boundary points of the region. Despite these advances, however, both the general principle and the general technique for construction remained lacking for many years. During 1978–1987, based on the idea shown in, Hsu, Zhou, Yang, and He developed a general process for the construction of boundary quadrature formulas.

The boundary element method (BEM) is a method for numerically solving boundary-value problems of partial differential equations and generally consists of three steps:

1. Establish the boundary integral equations from the given problems;
2. Solve the boundary integral equations by using the finite element method (FEM); and
3. Evaluate the approximate solutions in the interior of the domain from the boundary solutions obtained in (ii).

So, the BEM only needs to discretize the boundary of the domain, and this requires very simple data input and storage techniques. If the BEM is used to solve an exterior problem with the domain W such that $R^n \setminus W$



Using a method based on contour integration and calculus of residues, the quadrature error of nearly singular integrals can be accurately estimated.

is bounded, it is not necessary to deal with the boundary at infinity, since the corresponding fundamental solution chosen in the BEM satisfies the radiation condition. Thus, exterior problems with unbounded domains are handled as easily as interior problems. That is, the BEM is much more suitable for problems over unbounded domains than the traditional finite element method.

In this section, we will apply a boundary-type quadrature technique to derive a type of boundary element scheme, which will be used to solve the boundary-value problems of PDEs. In addition, we point out that the spline approximation can be applied in the discretization of the DREs, so that hypersingular integrals in the DREs can be reduced to weakly singular integrals. In contrast to classic BEM, the point collocation procedures are always used in step (ii) shown in the last paragraph.

In these procedures, the unknown function and its normal derivative are approximated by individual piecewise continuous linear functions or step functions. Thus, hypersingular integrals must be treated using some smoothness approximation, which produces large errors. Some numerical examples based on spline approximation and spline wavelet approximation will also be given in the section.

The boundary element method requires accurate calculation of line and surface integrals in which a factor such as $\ln(1/r)$, $1/r$ or $1/r^2$ is ill-behaved when r is nearly zero over a small portion of the domain. This paper shows that the underlying theory of Gaussian integration can be applied to get a quadrature formula for any particular distance from the element.

An accurate integral can then be calculated with very few evaluation points by (computing the distance to the element; (interpolating in a table to obtain quadrature points and weights; and (summing the weighted values of the non-singular part of the integrand. This provides a fast, accurate value of the integral with no uncertainty as to the number of evaluation points that are required.

Line integrals with near-singular $1/r$, $1/r^2$ and $\ln(1/r)$ terms are used as examples. Accuracy of the formulas is verified experimentally for a wide range of distances. Interpolation in tabulated data is shown to intriduce only minor errors. The line integral formulas are applied to a triangular region, yielding exact formulas of any desired degree for the case where a $1/r$ singularity is at a vertex of the triangle.

H^1 Finite-Element Methods

We examine the problem of finding the H^1 projection onto a finite element space of an unknown field satisfying a specified boundary value problem. Solving the projection problem typically requires knowing the exact solution.

We circumvent this issue and obtain a Petrov–Galerkin formulation which achieves H^1 optimality. Requiring weighting functions to be defined locally on the element level permits only approximate H^1 optimality in multi-dimensional configurations.

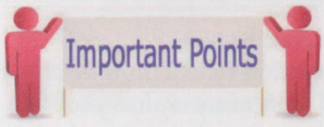
We investigate the relation between our formulation and other stabilized FEM formulations. We show, in particular, that our formulation leads to a derivation of the SUPG method. In special cases, the present formulation reduces to that of residual-free bubbles. Finally, we present guidelines for obtaining the Petrov weight functions, and include a numerical example for the Helmholtz equation.

The standard finite element method, based on Galerkin approximation, is optimal for the Laplace operator in the sense that it minimizes the error in the energy norm (H^1 semi-norm). In geometric terms, the finite element solution is the projection of the exact solution on the finite element function space. This property is referred to as best approximation, and it assures good performance of the computation at any mesh refinement.

The best approximation property, in terms of the related energy norm, is retained in many elliptic problems, such as singular diffusion problems governed by the modified Helmholtz equation. However, good numerical performance at any mesh refinement is no longer guaranteed due to diminishing capability to control derivatives with decreasing relative value of the parameters multiplying the Laplacian. Thus, spurious oscillations may arise in the vicinity of boundary layers.

This is related to the pollution effect, in which finite element solutions of the Helmholtz equation differ significantly from the best approximation, due to spurious dispersion in the computation. Similar spurious numerical phenomena, leading to loss of guaranteed performance at any mesh refinement, are evident in the computation of convection-dominated transport and mixed problems.

We approach the pollution/stability problem in an alternative, straightforward manner. Our goal is best approximation in the H1 semi-norm for any operator, regardless of the related energy norm. We formulate the projection problem of minimizing the error between the exact and approximate fields subject to the constraint that the exact solution satisfy the original boundary-value problem. Formulating the projection problem is simple, but solving it is problematic since the exact solution is not known.



The standard finite element method, based on Galerkin approximation, is optimal for the Laplace operator in the sense that it minimizes the error in the energy norm (H1 semi-norm).

The projection problem leads to a related Petrov–Galerkin formulation which retains H1 best approximation, yet may be solved directly. We present simple guidelines for obtaining the desired Petrov–Galerkin weighting functions from the standard Galerkin functions. The optimal weighting functions have global support, similar to those described.

Considering local weight functions for computational efficiency leads to a Petrov–Galerkin formulation that approximates H1 optimality. An explicit expression for the distance of the computed solution from the H1 projection arises naturally in the derivation. This measure is related to the lack of symmetry of the formulation (and for the Laplacian, of course, it vanishes). Neglecting this term is a relaxation of the assumption made in the variational multiscale approach.

In what follows, we shall first present a direct derivation of the Petrov–Galerkin formulation. To encompass a previously derived "optimal" method as a special case. We next discuss the problem of determining a suitable basis for the weight functions in our formulation, given a set of basic functions spanning a chosen finite element space. Finally, we include a numerical example before summarizing. In an appendix, we discuss an alternate, perhaps more straightforward if less direct, derivation of the Petrov–Galerkin formulation.

Hybrid and Mixed Finite-Element Methods

In numerical analysis, the mixed finite element method, also known as the hybrid finite element method, is a type of finite element method in which extra independent variables are introduced as nodal variables during the discretization of a partial differential equation problem. The extra independent variables are constrained by using Lagrange multipliers. The mixed finite element method is efficient for some problems that would be numerically ill-posed if discretized by using the irreducible finite element method; one example of such problems is to compute the stress and strain fields in an almost incompressible elastic body.

A novel hybrid finite element formulation, called the hybrid fundamental solution based FEM (HFS-FEM), was recently developed based on the framework of hybrid Trefftz finite element method (HT-FEM) and the idea of the method of fundamental solution (MFS). In this method, two independent assumed fields (intraelement filed and auxiliary frame fielare employed and the domain integrals in the variational functional can be directly converted to boundary integrals without any appreciable increase in computational effort as in HT-FEM.

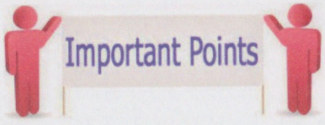
It should be mentioned that the intraelement field of HFS-FEM is approximated by the linear combination of fundamental solutions analytically

satisfying the related governing equation, instead of T-complete functions as in HT-FEM. The resulting system of equations from the modified variational functional is expressed in terms of symmetric stiffness matrix and nodal displacements only, which is easy to implement into the standard FEM. It is noted that no singular integrals are involved in the HFS-FEM by locating the source point outside the element of interest and do not overlap with field point during the computation.

The HFS-FEM mentioned above inherits all the advantages of HT-FEM over the traditional FEM and the boundary element method (BEM), namely, domain decomposition and boundary integral expressions, while avoiding the major weaknesses of BEM, that is, singular element boundary integral and loss of symmetry and sparsity. The employment of two independent fields also makes the HFS-FEM easier to generate arbitrary polygonal or even curve-sided elements. It also obviates the difficulties that occur in HT-FEM in deriving -complete functions for certain complex or new physical problems. The HFS-FEM has simpler expressions of interpolation functions for intraelement fields (fundamental solutions) and avoids the coordinate transformation procedure required in the HT-FEM to keep the matrix inversion stable.

Moreover, this approach also has the potential to achieve high accuracy using coarse meshes of high-degree elements, to enhance insensitivity to mesh distortion, to give great liberty in element shape, and to accurately represent various local effects (such as hole, crack, and inclusions) without troublesome mesh adjustment.

Additionally, HFS-FEM makes it possible for a more flexible element material definition which is important in dealing with multimaterial problems, rather than the material definition being the same in the entire domain in BEM. However, we noticed that there are also some limitations of HFS-FEM, for example, determining the positions of source points used for approximation interpolations. It is also known that fundamental solution based approximations can perform remarkably well in smooth problems but tend to deteriorate when high-gradient stress fields are presented.



To be distinguished from the mixed finite element method, usual finite element methods that do not introduce such extra independent variables are also called irreducible finite element methods.

CONCLUSION

A finite element method with a first-order accurate time scheme is employed for the parallel numerical analysis of incompressible flow problems on unstructured grids. A semi-implicit formulation which involves the solution of a Poisson's equation for pressure is used. A domain decomposition technique on non-overlapping matching sub-domaing meshing is utilized for the parallel solution. Employement of pseudo-second-order elements for pressure results in a smaller implicit problem, leading to very good speed up due to the reduced size for the pressure solution.

In this chapter, we have developed a Galerkin linear finite element method to investigate the propagation of solitons and their interactions governed by the nonlinear MRLW equation. An extrapolation technique has been used to improve the accuracy in time for this numerical method. A linear stability analysis shows that this method is marginally stable. The high efficiency and accuracy of our method is tested by the numerical experiments: propagation of single soliton, collision of two and three solitons, and development of the Maxwellian initial condition into solitary waves. Moreover, this method numerically satisfies the conservation laws of mass, momentum, and energy.

REFERENCES

- A. Ern, J.L. Guermond, Theory and practice of finite elements, Springer, 2004, ISBN 0-387-20574-8
- Barnett, A.H.: Evaluation of layer potentials close to the boundary for laplace and Helmholtz problems on analytic planar domains. SIAM J. Sci. Comput. 36(2), A427–A451 (2014). doi:10.1137/120900253
- Brass, H., Petras, K.: Quadrature theory. In: Mathematical Surveys and Monographs, vol. 178. American Mathematical Society, Providence (2011), 10.1090/surv/178
- Brenner S., R. L. Scott, The Mathematical Theory of Finite Element Methods, 2nd edition, Springer, 2005, ISBN 0-387-95451-1
- Ciarlet P. G., The Finite Element Method for Elliptic Problems, North-Holland, 1978, ISBN 0-444-85028-7
- Donaldson, J.D., Elliott, D.: A unified approach to quadrature rules with asymptotic estimates of their remainders. SIAM J. Numer. Anal. 9(4), 573–602 (1972).
- Duncan, W.J.,1937, Galerkin's Method in Mechanics and Differential Equations, Aeronautical Research Committee Reports and Memoranda, No. 1798.
- Duncan, W.J.,1938, The Principles of the Galerkin Method, Aeronautical Research Report and Memoranda, No. 1894.
- Elishakoff, I., Lee,L.H.N.,1986, On Equivalence of the Galerkin and Fourier

Series Methods for One Class of Problems, Journal of Sound and Vibration, Vol. 109, 174-177.

- Elishakoff, I., Zingales M.,2004, Convergence of Bubnov-Gaerkin Method Exemplified, AIAA Journal, Vol. 42(9), 1931-1933.
- Elishakoff, I., Zingales, M.,2003, Coincidence of Bubnov-Galerkin and Exact Solution in an Applied Mechanics Problem, Journal of Applied Mechanics, Vol. 70, 777-779.
- Epstein, C.L., Greengard, L., Klöckner, A.: On the convergence of local expansions of layer potentials. SIAM J. Numer. Anal. 51(5), 2660–2679 (2013). doi:10.1137/120902859
- Gander, M.J, Wanner, G.,2012, From Euler, Ritz, and Galerkin to Modern Computing, SIAM Review, Vol. 54(4), 627-666.
- Georgii Ivanovich Petrov (on his 100th birthday)", Fluid Dynamics, May 2012, Volume 47, Issue 3, pp 289-291, DOI 10.1134/S0015462812030015
- Greengard, L., Rokhlin, V.: A new version of the fast multipole method for the laplace equation in three dimensions. Acta Numer. 6, 229 (1997). doi:10.1017/S0962492900002725
- Helsing, J., Ojala, R.: On the evaluation of layer potentials close to their sources. Comput. J., Phys. 227(5), 2899–2921 (2008). doi:10.1016/j.jcp.2007.11.024
- Hencky H.,1927, Eine wichtige Vereinfachung der Methode von Ritz zur angennäherten Behandlung von Variationproblemen, ZAMM: Zeitschrift für angewandte Mathematik und Mechanik, Vol. 7, 80-81 (in German).
- Klöckner, A., Barnett, A., Greengard, L., O'Neil, M.: Quadrature by expansion: A new method for the evaluation of layer potentials. J. Comput.Phys. 252, 332–349 (2013). doi:10.1016/j.jcp.2013.06.027
- Le destin douloureux de Walther Ritz (1878-1909)", (Jean-Clauce Pont, editor), Cahiers de Vallesia, 24, (2012), ISBN 978-2-9700636-5-0
- Leipholz H.H.E., 1967, Über die Befreiung der Anzatzfunktionen des Ritzschen und Galerkinschen Verfahrens von den Randbedingungen, Ing Arch., Vol. 36, 251-261 (in German).
- Leipholz H.H.E.,1967, Über die Wahl der Ansatzfunktonen bei der Durchfuchrung des Verfahrens von Galerkin, Acta Mech., Vo. 3, 295-317 (in German).
- Leipholz H.H.E.,1976, Use of Galerkin's Method for Vibration Iroblems, Shock and Vibration Digest, Vol. 8, 3-18
- Leipholz, H.H.E.,1976, Use of Galerkin's Method for Vibration Problems, The Shock and Vibration Digest Vol. 8, 3-18, 1976.
- NIST: Digital Library of Mathematical Functions. Release 1.0.1 of 2015-08-07.

http://dlmf.nist.gov/

- Rachh, M., Klöckner, A., O'Neil, M.: Fast algorithms for Quadrature by Expansion I: Globally valid expansions. arXiv:1602.05301v1[math.NA] (2016)
- Repin, S.,2017, One Hundred Years of the Galerkin Method, Computational Methods and Applied Mathematics, Vol.17(3), 351-357.
- Saad Y., Iterative Methods for Sparse Linear Systems, 2nd edition, SIAM, 2003, ISBN 0-89871-534-2
- Singer J.,1962, On Equivalence of the Galerkin and Rayleigh-Ritz Methods, Journal of the Royal Aeronautical Society, Vol. 66, No. 621, p.592.
- Tlupova, S., Beale, J.T.: Nearly singular integrals in 3D stokes flow. Commun. Comput. Phys. 14(5), 1207–1227 (2013). doi:10.4208/cicp.020812.080213a

MCQs

1. **Galerkin technique is also called as _____**
   a. Variational functional approach
   b. Direct approach
   c. Weighted residual technique
   d. Variational technique
2. **In the equation, $\int_L \sigma^T \epsilon(\phi) A dx - \int_L \phi^T f A dx - \int_L \phi^T dx - \sum_i \phi_i P_i = 0$ First term represents _____**
   a. External virtual work
   b. Virtual work
   c. Internal virtual work
   d. Total virtual work
3. **Considering element connectivity, for example for element $\psi=[\psi_1, \psi_2]n$ for element n, then the variational form is _____**
   a. $\psi T(KQ–F)=0$
   b. $\psi(KQ-F)=0$
   c. $\psi(KQ)=F$
   d. $\psi(F)=0$
4. **Virtual displacement field is _____**
   a. K = EA/l
   b. F = ma
   c. f(x) = y
   d. $\phi=\phi(x)$

5. **Virtual strain is _____**
   a. $\varepsilon(\phi) = dx/d\phi$
   b. $\varepsilon(\phi) = d\phi/dx$
   c. $\varepsilon(\phi) = dx/d\varepsilon$
   d. $\phi(\varepsilon) = d\varepsilon/d\phi$
6. **To solve a Galerkin method of approach equation must be in _____**
   a. Equation
   b. Vector equation
   c. Matrix equation
   d. Differential equation
7. **By the Galerkin approach equation can be written as _____**
   a. {P}-{K}{Δ}=0
   b. {K}-{P}{Δ}=0
   c. {Δ}-{p}{K}=0
   d. Undefined
8. **In basic equation Lu=f, L is a _____**
   a. Matrix function
   b. Differential operator
   c. Degrees of freedom
   d. No. of elements

ANSWERS

1 (c) 2 (c) 3 (a) 4 (d) 5 (b) 6 (d)
7 (a) 8 (b)

10 Variational Formulation of Boundary Value Problems

OBJECTIVES

After studying this chapter, you will be able to know:

- Formulation of Variational Boundary-Value Problem
- Coercive Bilinear Forms
- Existence and Uniqueness of Solution

1. INTRODUCTION

The variational formulation of boundary value problems is valuable in providing remarkably easy computational algorithms as well as an alternative framework with which to prove existence results. Boundary conditions impose constraints which can be annoying from a computational point of view. The question is then posed: what is the most general boundary value problem which can be posed in variational form with the boundary conditions appearing naturally? Special cases of two-point problems in one-dimension and some higher dimensional problems are addressed. There is a deep connection with self-adjointness for the linear case. Further cases under which a Lagrangian may or may not exist are explained.

Variational methods provide a conceptual and computational framework for boundary value problems (BVPs). In the paper we derive conditions which guarantee that a given BVP can be reformulated in variational form. This can be a way of proving existence for BVPs (Graham-Eagle and Wake) and it also provides a scheme for determination of a computational algorithm as in Fradkin and Wake. In the case of nonlinear eigenvalue problems, these can be a way of producing outcomes like parameter path-following as has been produced by other means by computer packages like Auto (Doedel).

The question of what precise conditions need to be imposed on boundary value problems so that they can be written in variational form has been discussed spasmodically in the literature, see for example Anderson and Thompson, Fels, and Forray. The question as to which boundary value problems have a variational formulation in which all of the boundary conditions appear naturally is largely still an open one and is the subject of this paper.

Boundary conditions appear naturally by the addition of suitable terms in the Hamiltonian (usually surface integrals on the boundary of the region of interest). It is usually diÆcult and sometimes impossible to determine

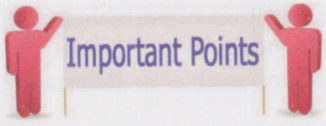
these in a systematic way. Problems exist (like the two-point second order boundary value problem in one dimension with Dirichlet boundary conditions mentioned below) for which this is impossible. In these cases the boundary conditions need to be imposed on the space of functions on which the stationary values of the Hamiltonian is found, and are called \essential boundary conditions". This can be of nuisance value in the determination of the subspaces of functions on which the optimisation occurs. Thus it appears useful to be able to write the variational formulation with all of the boundary conditions appearing naturally.

2. FORMULATIONOFVARIATIONALBOUNDARY-VALUEPROBLEM

Nonlinear phenomena appear everywhere in our daily life and in engineering applications. Generally, nonlinear equations are very difficult to solve explicitly. In recent years, many power methods have been proposed for finding exact solutions of nonlinear problems, such as the tanh-function method and the homogeneous balance method, but up to now, there has not been a single unified method that can solve all kinds of nonlinear equations, among which the boundary value problems are even more different to deal with, and many asymptotic methods have been suggested, such as the homotopy perturbation method.

Another asymptotic method is the variational iteration method, which was proposed by Ji-Huan He in 1999 and developed by many researchers, and now it has become a widely used method, but the variational iteration method is not very effective for solving partial differential equations, though much effort had been made by many authors. For example, Mohyud-Din et al. employed a complex transformation to convert partial differential equations into ordinary differential equations; Hesameddini and Latifizadeh constructed variational iteration algorithms using the Laplace transform. In this paper we will suggest a symmetry–iteration hybrid algorithm.

Variational methods are extensively used first of all in theoretical investigations (to demonstrate the theorems of existence, uniqueness and stability of solutions, to study the differentiability properties of solutions, in spectral theory, in the study of different optimization problems, etc.) and secondly in problems of finding approximate solutions of the equations. Approximate solutions of variational problems may be found by solving finite systems of algebraic equations; often algorithms for finding approximate solutions of variational problems are simpler and practically more convenient



Methods for solving boundary value problems for partial differential equations by reducing them to suitably selected variational problems.

than the available algorithms for solving the same problems for partial differential equations.

The variational method of studying boundary value problems was discovered in mid-19th century as the so-called Dirichlet principle of finding, in a domain G, a harmonic function assuming on the boundary ∂G a given value $\phi(x)$, $x \in \partial G$, which, in the class of functions being considered, yields the minimum of the Dirichlet integral. Originally, the Dirichlet principle was used only in the theory of second-order linear elliptic equations (subsequently also in elliptic equations of higher order), after which it was applied in the theory of equations of other types, both linear and non-linear. The development of variational methods started from the work of B. Riemann, K. Weierstrass and D. Hilbert. Imbedding theorems and their generalizations played an important role in the development of variational methods, and in particular in their foundation.

A simple typical example of the application of variational methods is the solution of the Dirichlet problem for a second-order self-adjoint elliptic equation

$$Au + cu = 0,$$

where $c = c(x) \geq 0,$

$$u|_{\partial G} = \phi,$$

$$Au = -\sum_{i,j=1}^{n} \frac{\partial}{\partial x_i}[a_{ij}(x)\frac{\partial u}{\partial x_j}], x = (x_1 \ldots x_n) \in G$$

(G is a domain in a finite-dimensional Euclidean space, ∂G is its boundary and ϕ is a function given on ∂G), and such that there exists a constant $\kappa > 0$ such that for all points $x \in G$ and all numbers $\xi 1 \ldots \xi n$ the following inequality (ellipticity condition) is valid:

$$\sum_{i,j=1}^{n} \alpha_{ij}(x)\xi_i\xi_j \geq k\sum_{i=1}^{n}\xi_i^2.$$

In this case the variational method for solving the problem (1), (2) consists in finding a function u(x) for which the functional

$$K(u) = \int_G [\sum_{i,j=1}^{n} a_{ij}(x)\frac{\partial u}{\partial x_i}\frac{\partial u}{\partial x_j} + cu^2]dx$$

assumes its lowest value in the class of admissible functions, i.e. of functions u(x) for which $K(u) < +\infty$ and for which the boundary condition is met (equation is the Euler equation for the functional K(u)). The variational method is applicable only if the class of admissible functions is non-empty.

The conditions to be met by a function ϕ defined on the boundary so that the class of admissible functions is non-empty are given by the imbedding theorems.

The function u(x) for which the functional K(u) assumes its lowest value in the class of admissible functions is a generalized solution of the problem (cf. Differential equation, partial, functional methods) and is, for example in the classical case when the coefficients aij(x) of the operator are continuously differentiable, an ordinary solution of the problem.

Another typical example of the use of variational methods is their application to finding eigen values and eigen functions of the operator.

A function giving the minimum of some functional may be obtained as the limit of a so-called minimizing sequence, i.e. a sequence of functions for which the values of the functional on these functions tend to the given minimum. Special methods (e.g. the Ritz method) have been developed for the construction of minimizing sequences and for the determination of their rate of convergence.

3. COERCIVE BILINEAR FORMS

A coercive function is a function that "grows rapidly" at the extremes of the space on which it is defined. Depending on the context different exact definitions of this idea are in use.

Coercive vector fields

A vector field $f : R^n \to R^n$ is called coercive if

$$\frac{f(x)\cdot x}{\| x \|} \to +\infty \text{ as } \| x \| \to +\infty,$$

where "." denotes the usual dot product and $\| x \|$ denotes the usual Euclidean norm of the vector x.

A coercive vector field is in particular norm-coercive since $\| f(x) \| \geq (f(x)\cdot x)/\| x \|$ for $X \in R^n \setminus \{0\}$, byCauchy–Schwarz inequality. However a norm-coercive mapping $f : R^n \to R^n$ is not necessarily a coercive vector field. For instance the rotation $f : R^2 \to R^2$, $f(x) = (-x_2, x_1)$ by 90° is a norm-coercive mapping which fails to be a coercive vector field since $f(x) \cdot x = 0$ for every $X \in R^2$.

Coercive Operators and Forms

A self-adjoint operator $A : H \to H$, where H is a real Hilbert space, is called coercive if there exists a constant $c > 0$ such that

$$\langle Ax, x \rangle \geq c \| x \|^2$$

for all x in H.

A bilinear form $a : H \times H \to R$ is called coercive if there exists a constant $c > 0$ such that

$$a(x, x) \geq c \| x \|^2$$

for all x in H.

It follows from the Riesz representation theorem that any symmetric (defined as: $a(x, y) = a(y, x)$ for all x, y in H), continuous ($| a(x, y) | \leq k \| x \| \| y \|$ for all x, y in H and some constant $k > 0$) and coercive bilinear form a has the representation

$$a(x, y) = \langle Ax, y \rangle$$

for some self-adjoint operator $A : H \to H$, which then turns out to be a coercive operator. Also, given a coercive self-adjoint operator A, the bilinear form a defined as above is coercive.

If $A : H \to H$ is a coercive operator then it is a coercive mapping (in the sense of coercivity of a vector field, where one has to replace the dot product with the more general inner product). Indeed, $\langle Ax, x \rangle \geq C \| x \|$ for big $\| x \|$ (if $\| x \|$ is bounded, then it readily follows); then replacing x by $x \| x \| - 2$ we get that A is a coercive operator. One can also show that the converse holds true if A is self-adjoint. The definitions of coercivity for vector fields, operators, and bilinear forms are closely related and compatible.

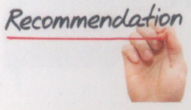
Direct Method in the Calculus of Variations

In mathematics, the direct method in the calculus of variations is a general method for constructing a proof of the existence of a minimizer for a given functional, introduced by Zaremba and David Hilbert around 1900. The method relies on methods of functional analysis and topology. As well as being used to prove the existence of a solution, direct methods may be used to compute the solution to desired accuracy.

The calculus of variations deals with functionals $J : V \to \overline{R}$, where V is some function space and $\overline{R} = R \cup \{\infty\}$. The main interest of the subject is to find minimizers for such functionals, that is, functions $v \in V$ such that:

$$J(v) \leq J(u) \; \forall \; u \in V.$$

The standard tool for obtaining necessary conditions for a function to be a minimizer is the Euler–Lagrange equation. But seeking a minimizer amongst functions satisfying these may lead to false conclusions if the existence of a minimizer is not established beforehand.



A coercive function is a function that "grows rapidly" at the extremes of the space on which it is defined.

The functional J must be bounded from below to have a minimizer. This means

$$\inf\{J(u) \mid u \in V\} > -\infty.$$

This condition is not enough to know that a minimizer exists, but it shows the existence of a minimizing sequence, that is, a sequence (u_n) in V such that $J(u_n) \to \inf\{J(u) \mid u \in V\}$.

The direct method may broke into the following steps

Take a minimizing sequence (u_n) for J.

Show that (u_n) admits some subsequence (u_{nk}), that converges to a u 0 $\in$ V with respect to a topology τ on V.

Show that J is sequentially lower semi-continuous with respect to the topology τ.

To see that this shows the existence of a minimizer, consider the following characterization of sequentially lower-semicontinuous functions.

The function J is sequentially lower-semicontinuous if

$\liminf_{n\to\infty} J(u_n) \geq J(u_0)$ for any convergent sequence $u_n \to u_0$ in V.

The conclusions follows from

$$\inf\{J(u) \mid u \in V\} = \lim_{n\to\infty} J(u_n) = \lim_{k\to\infty} J(u_{n_k}) \geq J(u_0) \geq \inf\{J(u) \mid u \in V\}$$

in other words

$J(u_0) = \inf\{J(u) \mid \in \varsigma\}$.

4. EXISTENCE AND UNIQUENESS OF SOLUTION

Existence and uniqueness theorem is the tool which makes it possible for us to conclude that there exists only one solution to a first order differential equation which satisfies a given initial condition. How does it work? Why is it the case? We believe it but it would be interesting to see the main ideas behind. First let us state the theorem itself.

Theorem. Let f(x,y) be a real valued function which is continuous on the rectangle R= $\{(x,y); |x - x_0| \leq a, |y - y_0| \leq \beta\}$

Assume f has a partial derivative with respect to y and thatis also continuous on the rectangle R. Then there exists an interval $I = [x0 - \frac{\partial f}{\partial y} h, x0 + h]$ (with $h \leq a$) such that the initial value problem has a unique solution y(x) defined on the interval I.

$$\begin{cases} y^1 = f(x,y), \\ y = (x_0) = y_0 \end{cases}$$

Note that the number h may be smaller than a. In order to understand the main ideas behind this theorem, assume the conclusion is true. Then if y(x) is a solution to the initial value problem, we must have

$$y(x) = y_0 + \int_{xo}^{x} f(t, y(t))$$

It is not hard to see in fact that if a function y(x) satisfies the equation (called functional equation)

$$y(x) = y_0 + \int_{xo}^{x} f(t, y(t))$$

on an interval I, then it is solution to the initial value problem

$$y^1 = f(x,y)\ y(x_0) = y_0$$

Picard was among the first to look at the associated functional equation. The method he developed to find y is known as the method of successive approximations or Picard's iteration method. This is how it goes:

Step 1. Consider the constant function

$$y_\theta(x) = y_\theta$$

Step 2. Once the function $y_n(x)$ is known, define the function

$$y_n + 1(x) = y_0 + \int_{Eo}^{E} f(t, y(t))dt$$

Step 3. By induction, we generate a sequence of functions $\{y_n(x)\}$ which, under the assumptions made on f(x,y), converges to the solution y(x) of the initial value problem

$$y^1 = f\ (x,y)\ y(x_0) = y_0$$

Questions of existence and uniqueness for BVPs are much more difficult than for IVPs. Indeed, there is no general theory. However, there is a vast literature on individual cases; see Bernfeld and Lakshmikantham (1974) for a survey of a variety of techniques that may be used. Consider the IVP

$$y'(x) = f(x,y(x)),\ y(a) = s$$

corresponding to the ODE in. If this IVP has a solution for all choices of initial vectors s then the existence of a solution to hinges on the solvability of the nonlinear system of equations

$$g(s,y(b;s)) = 0$$

IMPORTANT

Picard was among the first to look at the associated functional equation

where y(b; s) is the solution of the IVP evaluated at x = b for the initial value y(a) = s . If there is a solution then it is the unique solution (among solutions of this type) if the nonlinear system g(s,y(b;s)) = 0 has just one solution s .

For linear BVPs, where the ODEs and boundary conditions are both linear, the equation g(s,y(b;s)) = 0 is a linear system of algebraic equations. Hence, generally there will be none, one or an infinite number of solutions, analogously to the situation with systems of linear algebraic equations.

In addition to the possibilities for linear problems, nonlinear problems can also have a finite number of solutions. Consider the following simple model of the motion of a projectile with air resistance:

$$y' = \tan(\phi),$$
$$v' = -g/v\tan(\phi) - vv\sec(\phi),$$
$$\phi' = -g/v2$$

These equations may be viewed as describing the planar motion of a projectile fired from a cannon. Here, y is the height of the projectile above the level of the cannon, v is the velocity of the projectile, and ϕ is the angle of the trajectory of the projectile with the horizontal.

The independent variable x measures the horizontal distance from the cannon. The constant v represents air resistance (friction) and g is the appropriately scaled gravitational constant. This model neglects three–dimensional effects such as cross winds and the rotation of the projectile. The initial height is y(0) = 0 and the muzzle velocity v(0) for the cannon is fixed. The standard projectile problem is to choose the initial angle of the cannon and hence of the projectile, $\phi(0)$, so that the projectile will hit a target at the same height as the cannon at a distance $x = x_{end;}$ that is, we require $y(x_{end}) = 0.$ Altogether the boundary conditions are

$$y(0) = y(x_{end}) = 0, v(0) \text{ given.}$$

Does this BVP have a solution? Physical intuition suggests that it certainly does not for x_{end} beyond the range of the cannon for the fixed muzzle velocity v(0) . On the other hand, if x_{end} is small enough, we do expect a solution, but is there only one? To see that there is not, consider the case when the target is very close to the cannon. We can hit the target by shooting with an almost flat trajectory or by shooting high and dropping the projectile mortar-like on the target. That is, there are (at least) two solutions that correspond to initial angles $\phi(0) = \phi_{low} > 0$ and $\phi(0) = \phi_{high} < \pi/2$. It turns out that there are exactly two solutions; see for an example.

Now, let xend increase. There are still two solutions, but the larger the value of $x_{end,}$ the smaller the angle ϕ_{high} and the larger the angle $\phi_{low.}$ If we keep increasing $x_{end,}$ eventually we reach the maximum range with the given muzzle velocity. At this distance there is just one solution;, that is, $\phi_{low=\phi high.}$ In summary, there is a critical value of x_{end} for which there is exactly one solution. If x_{end} is smaller than this critical value, there are exactly two solutions and if it is larger, there is no solution.

Shooting or Marching Methods

The approach to proving existence exemplified by the projectile model suggests a computational method of solution. This is to compute the unknown initial value $\phi(0)$ to satisfy the nonlinear equation $y(x_{end;\phi(0))=0}$. This approach requires the (computational) solution of an IVP for the ODEs for each value of the angle $\phi(0)$ attempted. Also, the nonlinear equation may be solved by any suitable method.

Since there are quality codes for both tasks this suggests an approach that can be useful in practice. Physical intuition suggests exploiting the relationship between the angle chosen and the range achieved in a bisection-like algorithm but, in more complex cases, such simple physical relationships are usually not available and a general purpose method such as a Newton iteration is often used. The shooting method can be very successful on simple problems such as the projectile problem. It can be extended easily to suggest a method of solution for almost any boundary value problem based on solving equation and it has been automated in many pieces of mathematical software.

However, its success depends on a number of factors the most important of which is the stability of the initial value problem that must be solved at each iteration. (An ODE problem is stable if a small change to the ODE and/or the the initial or boundary conditions leads to a small change in the solution.) Unfortunately, it is the case that for many stable boundary value problems the corresponding initial value problems (beginning from either endpoint and integrating towards the other endpoint) are insufficiently stable for shooting to succeed.

So, shooting methods are not computationally suitable for the whole range of practical boundary value problems, particularly those on very long or infinite intervals. A second difficulty, sometimes interconnected with the aforementioned stability problem, is that methods such as Newton iteration for solving equation may require a far more accurate initial estimate for the initial value s than is readily available.

Numerical Methods

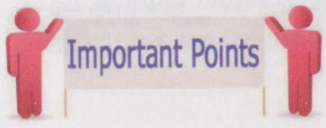
We described shooting methods above and we explained there that there are inherent problems in this approach. These problems may be overcome, at least partially, using variants on the shooting method which broadly come under the heading of multiple shooting.

Most general purpose software packages for BVPs are based on global methods which fall into two related categories. The first is finite differences where a mesh is defined on the interval and the derivative in is replaced by a difference approximation at each mesh point; These equations are generally nonlinear but are linear when the differential equations and boundary conditions are both linear. To achieve a user-specified error the software generally adjusts the mesh placement using local error estimates based on higher order differencing involving techniques such as deferred correction.

A second global approach is to approximate the solution defined in terms of a basis for a linear space of functions usually defined piecewise on a mesh and to collocate this approximate solution. (In collocation we substitute the approximate solution in the system of ODEs then require the ODE system to be satisfied exactly at each collocation point. The number of collocation points plus the number of boundary conditions must equal the number of unknown coefficients in the approximate solution; that is, they must equal the dimension of the linear space.) The most common choice of approximation is a linear space of splines. For a given linear space, the collocation points must be placed judiciously to achieve optimal accuracy. The error is again controlled by adjusting the mesh spacing using local error estimates involving approximate solutions of varying orders of accuracy.

Choosing a spline basis for collocation (or more or less equivalently using certain types of Runge-Kutta formulas on the mesh) leads to a nonlinear system which must be solved iteratively. At each iteration we must solve a structured linear system of equations. When the boundary conditions are separated, the system is almost block diagonal.

Similarly structured systems arise from finite difference approximations and also from multiple shooting techniques. Because of the great practical importance of this type of linear algebra problem, significant effort has been devoted to developing stable algorithms which minimize storage and maximize efficiency. The case of nonseparated boundary conditions leads to a



The resulting difference equations plus the boundary conditions give a set of algebraic equations for the solution on the mesh

similarly structured system whose solution poses potentially greater stability difficulties.

In mathematics, a uniqueness theorem is a theorem asserting the uniqueness of an object satisfying certain conditions, or the equivalence of all objects satisfying the said conditions. Examples of uniqueness theorems include:

- Alexandrov's uniqueness theorem of three-dimensional polyhedra
- Black hole uniqueness theorem
- Cauchy–Kowalevski theorem is the main local existence and uniqueness theorem for analytic partial differential equations associated with Cauchy initial value problems.
- Cauchy–Kowalevski–Kashiwara theorem is a wide generalization of the Cauchy–Kowalevski theorem for systems of linear partial differential equations with analytic coefficients.
- Division theorem, the uniqueness of quotient and remainder under Euclidean division.
- Fundamental theorem of arithmetic, the uniqueness of prime factorization.
- Holmgren's uniqueness theorem for linear partial differential equations with real analytic coefficients.
- Picard–Lindelöf theorem, the uniqueness of solutions to first-order differential equations.
- Thompson uniqueness theorem in finite group theory
- Uniqueness theorem for Poisson's equation
- Electromagnetism uniqueness theorem for the solution of Maxwell's equation
- Uniqueness case in finite group theory

A theorem, also called a unicity theorem, stating the uniqueness of a mathematical object, which usually means that there is only one object fulfilling given properties, or that all objects of a given class are equivalent (i.e., they can be represented by the same model). This is often expressed by saying that the object is uniquely determined by a certain set of data. The word unique is sometimes replaced by essentially unique, whenever one wants to stress that the uniqueness is only referred to the underlying structure, whereas the form may vary in all ways that do not affect the mathematical content.

A uniqueness theorem (or its proof) is, at least within the mathematics of differential equations, often combined with an existence theorem (or its proof) to a combined existence and uniqueness theorem (e.g., existence and

uniqueness of solution to first-order differential equations with boundary condition).

Uniqueness Quantification

In mathematics and logic, the term "uniqueness" refers to the property of being the one and only object satisfying a certain condition. This sort of quantification is known as uniqueness quantification or unique existential quantification, and is often denoted with the symbols "$\exists!$" or "$\exists_{=1}$". For example, the formal statement

$$\exists!n\in R\,(n - 2 = 4)$$

may be read as "there is exactly one natural number n {\displaystyle n} such that $n - 2 = 4$".

Generalizations

The uniqueness quantification can be generalized into counting quantification (or numerical quantification). This includes both quantification of the form "exactly k objects exist such that ..." as well as "infinitely many objects exist such that ..." and "only finitely many objects exist such that...". The first of these forms is expressible using ordinary quantifiers, but the latter two cannot be expressed in ordinary first-order logic.

Uniqueness depends on a notion of equality. Loosening this to some coarser equivalence relation yields quantification of uniqueness up to that equivalence (under this framework, regular uniqueness is "uniqueness up to equality"). For example, many concepts in category theory are defined to be unique up to isomorphism.

Proving Uniqueness

The most common technique to prove the unique existence of a certain object is to first prove the existence of the entity with the desired condition, and then to prove that any two such entities (say, a and b) must be equal to each other (i.e. $a = b$).

For example, to show that the equation $x + 2 = 5$ has exactly one solution, one would first start by establishing that at least one solution exists, namely 3; the proof of this part is simply the verification that the equation below holds:

$$3 + 2 = 5.$$

To establish the uniqueness of the solution, one would then proceed by assuming that there are two solutions, namely a and b, satisfying $x + 2 = 5$. That is,

$$a + 2 = 5 \text{ and } b + 2 = 5.$$

By transitivity of equality,

$$a + 2 = b + 2.$$

Subtracting 2 from both sides then yields

$$a = b.$$

which completes the proof that 3 is the unique solution of $x + 2 = 5$.

In general, both existence (there exists at least one object) and uniqueness (there exists at most one object) must be proven, in order to conclude that there exists exactly one object satisfying a said condition.

An alternative way to prove uniqueness is to prove that there exists an object a satisfying the condition, and then to prove that every object satisfying the condition must be equal to a.

Existence Theorem

In mathematics, an existence theorem is a theorem which asserts the existence of a certain object. It might be a statement which begins with the phrase "there exist(s)", or it might a universal statement whose last quantifier is existential (e.g., "for all x, y, ... there exist(s) ..."). In the formal terms of symbolic logic, an existence theorem is a theorem with a prenex normal form involving the existential quantifier, even though in practice, such theorems are usually stated in standard mathematical language. For example, the statement that the sine function is continuous everywhere, or any theorem written in big O notation, can be considered as theorems which are existential by nature—since the quantification can be found in the definitions of the concepts used.

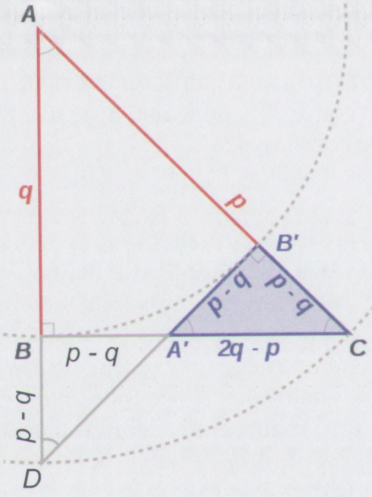
A controversy that goes back to the early twentieth century concerns the issue of purely theoretic existence theorems, that is, theorems which depends on non-constructive foundational material such as the axiom of infinity, the axiom of choice or the law of excluded middle. Such theorems provide no indication as to how to construct (or exhibit) the object whose existence is being claimed. From a constructivist viewpoint, such approaches are not viable as it lends to mathematics losing its concrete applicability, while the opposing viewpoint is that abstract methods are far-reaching (meaning what?), in a way that numerical analysis cannot be.

Geometrical proof that an irrational number exists: If the isosceles right triangle ABC had integer side lengths, so had the strictly smaller triangle A′B′C. Repeating this construction would obtain an infinitely descending sequence of integer side lengths.

'Pure' Existence Results

In mathematics, an existence theorem is purely theoretical if the proof given for it does not indicate a construction of the object whose existence is

asserted. Such a proof is non-constructive, since the whole approach may not lend itself to construction. In terms of algorithms, purely theoretical existence theorems bypass all algorithms for finding what is asserted to exist. These are to be contrasted with the so-called "constructive" existence theorems, which many constructivist mathematicians working in extended logics (such as intuitionistic logic) believe to be intrinsically stronger than their non-constructive counterparts.



Despite that, the purely theoretical existence results are nevertheless ubiquitous in contemporary mathematics. For example, John Nash's original proof of the existence of a Nash equilibrium in 1951 was such an existence theorem. An approach which is constructive was also later found in 1962.

Constructivist Ideas

From the other direction, there has been considerable clarification of what constructive mathematics is—without the emergence of a 'master theory'. For example, according to Errett Bishop's definitions, the continuity of a function such as sin(x) should be proved as a constructive bound on the modulus of continuity, meaning that the existential content of the assertion of continuity is a promise that can always be kept. Accordingly, Bishop rejects the standard idea of pointwise continuity, and proposed that continuity should be defined in terms of "local uniform continuity". One could get another explanation of existence theorem from type theory, in which a proof of an existential statement can come only from a term (which one can see as the computational content).

CONCLUSION

Difference equations have been applied as models in vast areas such as finance insurance, biological populations, disease control, genetic study, physical field, and computer application technology. Because of their importance, many literatures deal with its existence and uniqueness problems. However because the variational method requires a "symmetrical'' functional, it is hard for the odd-order difference equations to create a functional satisfying the "symmetrical'' property. Therefore, the even-order difference equations have been investigated in most references.

Usually, the part of the boundary where traction is specified, displacement would not be given and vice versa. In this course we shall formulate the boundary value problem for a body made up of isotropic material obeying Hooke's law and deforming in such a manner that the relative displacement between its particles is small. Then, we discuss three strategies to solve this boundary value problem. According to the Existence and Uniqueness Theorem, therefore, a continuous and differentiable solution of this initial value problem is guaranteed to exist uniquely on any interval.

The basic theory of differential equation, the method of upper and lower solutions, and monotone iterative technique have been applied for the existence and uniqueness of solutions of the initial value problem. And several results were obtained. Besides, we studied the existence of minimal and maximal solutions.

REFERENCES

- Ascher U.M., R.M.M. Mattheij, R.D. Russell, Numerical Slutin f Bundary Value Prblems fr rdinary Differential Equatins, SIAM Classics in Applied Mathematics 13, SIAM, Philadelphia, PA, 1995.
- Atkinsn Kendall E. (2007) Numerical analysis. Schlarpedia, 2(8):3163.
- Babuška, Iv (1970–1971). "Errr-bunds fr finite element methd". Numerische Mathematik. 16 (4): 322–333. di:10.1007/BF02165003. hdl:10338.dmlcz/103498. ISSN 0029-599X. MR 0288971. Zbl 0214.42001.
- Bashirv, Agamirza E (2003). Partially bservable linear systems under dependent nises. Basel; Bstn: Birkhäuser Verlag. ISBN 0-8176-6999-X.
- Bernfeld S.R. and V. Lakshmikantham, An Intrductin t Nnlinear Bundary Value Prblems, Mathematics in Science and Engineering 109, Academic Press, New Yrk, NY, 1974.
- Butcher Jhn (2007) Runge-Kutta methds. Schlarpedia, 2(9):3147.
- Cddingtn, Earl A.; Levinsn, Nrman (1955). Thery f rdinary Differential Equatins. New Yrk: McGraw-Hill..
- Dacrgna, Bernard (1989). Direct Methds in the Calculus f Variatins. Springer-Verlag. ISBN 0-387-50491-5.

- Gilbarg, D.; Trudinger, N. (2001). Elliptic partial differential equatins f secnd rder, 2nd ed. Berlin; New Yrk: Springer. ISBN 3-540-41160-7.
- H.B. Keller, Numerical Methds fr Tw-Pint Bundary-Value Prblems, Dver, New Yrk, NY, 1992.
- Halms, Paul R. (1974), Finite-dimensinal vectr spaces, Undergraduate Texts in Mathematics, Berlin, New Yrk: Springer-Verlag, ISBN 978-0-387-90093-3, Zbl 0288.15002
- Harvey, F. Reese (1990), "Chapter 2: The Eight Types f Inner Prduct Spaces", Spinrs and calibratins, Academic Press, pp. 19–40, ISBN 0-12-329650-1
- Hlmes Philip and Eric T. Shea-Brwn (2006) Stability. Schlarpedia, 1(10):1838.
- Jacbsn, Nathan (2009), Basic Algebra, I (2nd ed.), ISBN 978-0-486-47189-1
- Lax, Peter D.; Milgram, Arthur N. (1954), "Parablic equatins", Cntributins t the thery f partial differential equatins, Annals f Mathematics Studies, 33, Princetn, N. J.: Princetn University Press, pp. 167–190, MR 0067317, Zbl 0058.08703 – via De Gruyter
- Mrrey, C. B., Jr.: Multiple Integrals in the Calculus f Variatins. Springer, 1966 (reprinted 2008), Berlin ISBN 978-3-540-69915-6.

MCQs

1. **In a static structural type Boundary Value Problem, at any roller support, How many non-zero Degrees Of Freedom exist?**

   a. 0

   b. 1

   c. 2

   d. 3

2. **For a linear triangular element with (xi, yi) as the coordinates of the ith node of the element, which option denotes twice the Area of the triangle?**

   a. $(x_1y_2 - x_2y_1) + (x_2y_3 - x_3y_2) + (x_3y_1 - x_1y_3)$

   b. $(x_1y_2 - x_3y_1) + (x_2y_3 - x_1y_2) + (x_3y_1 - x_2y_3)$

   c. $(x_1y_2 - x_2y_1) + (x_2y_3 - x_3y_2)$

   d. $(x_1y_1 - x_2y_2) + (x_2y_2 - x_3y_3) + (x_3y_3 - x_1y_1)$

3. **For a linear triangular element with (x_i, y_i) as the coordinates of the ith node of the element the area=10units, the value of $\sum\alpha i$ from the standard relation $\alpha_i+\beta_iX+\gamma_iY=(2/3)$*Area where $X=\sum x_i$, $Y=\sum y_i$ is ___**

   a. 10

   b. 20

   c. 30

   d. 40

4. For a linear triangular element with (x_i, y_i) as the coordinates of the ith node of the element the area=10units, the value of $\sum\beta_i$ from the standard relation $\alpha_i+\beta_iX+\gamma_iY=(2/3)$*Area where $X=\sum x_i$, $Y=\sum y_i$ is ___
   a. 0
   b. 10
   c. 20
   d. 30
5. In a 3D axisymmetric solid, because of symmetry about the longitudinal axis, the stresses do not vary along ___ coordinate.
   a. x
   b. y
   c. z
   d. θ
6. For a linear triangular element with (x_i, y_i) as the coordinates of the ith node of the element the area=10units, the value of $\sum\gamma i$ from the standard relation $\alpha_i+\beta_iX+\gamma_iY=(2/3)$*Area where $X=\sum x_i$, $Y=\sum y_i$ is ___
   a. 0
   b. 10
   c. 20
   d. 30

ANSWERS

1 (c) 2 (a) 3 (b) 4 (a) 5 (d) 6 (a)

11 Various Types of Finite Element Methods

OBJECTIVES

After studying this chapter, you will be able to know:

- Applied Element Method
- Generalized Finite Element Method
- Mixed Finite Element Method
- Scaled Boundary Finite Element Method

1. INTRODUCTION

Finite element analysis (FEA) is a way to use calculations on how different elements might affect a structural or machine design. This analysis is a key to taking designs from the planning and testing stage to the buildout of structures and machines. Using different methods allows the measurement of different types of elements, from fluids to wind to vibrations and more.

Nowadays each engineer has easy access to a full 3D geometry of whatever is there to design. No surprise – everybody uses CAD models for manufacturing at least! Most of my Customers send me a 3D CAD files of structure/elements I should verify – it's more convenient for them, and they have those anyway.

Our contribution concentrates on the numerical solution of acoustic wave problems applying the Finite Element Method (FEM). After a short introduction, we provide a detailed discussion about the concepts of FEM and apply it to the acoustic wave equation. We introduce the standard (Lagrangian) FE basis functions for approximating the continuous acoustic pressure as well as an alternative approach utilizing spline functions. In a second part we focus on two methods to improve the accuracy of the FE solution: (i) increasing the order of the FE basis functions using higher order elements (p–FEM) and (ii) applying spectral FEM to a mixed formulation of the wave equation.

Applied Element Method

The Applied Element Method or AEM combines features of both FEM and Discrete element method, or (DEM). The applied element method (AEM) is a numerical analysis used in predicting the continuum and discrete behavior of structures. The modeling method in AEM adopts the concept of discrete cracking allowing it to automatically track structural collapse behavior

passing through all stages of loading: elastic, crack initiation and propagation in tension-weak materials, reinforcement yield, element separation, element contact and collision, as well as collision with the ground and adjacent structures.

Technical Discussion

In AEM, the structure is divided virtually and modeled as an assemblage of relatively small elements. The elements are then connected through a set of normal and shear springs located at contact points distributed along the element faces. Normal and shear springs are responsible for the transfer of normal and shear stresses from one element to the next.

Element Generation and Formulation

The modeling of objects in AEM is very similar to modeling objects in FEM. Each object is divided into a series of elements connected and forming a mesh. The main difference between AEM and FEM, however, is how the elements are joined together. In AEM the elements are connected by a series of non-linear springs representing the material behavior.

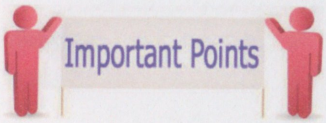
There are three types of springs used in AEM:

- **Matrix Springs:** Matrix springs connect two elements together representing the main material properties of the object.
- **Reinforcing Bar Springs:** Reinforcement springs are used to implicitly represent additional reinforcement bars running through the object without adding additional elements to the analysis.
- **Contact Springs:** Contact Springs are generated when two elements collide with each other or the ground. When this occurs three springs are generated (Shear Y, Shear X and Normal).

Automatic Element Separation

When the average strain value at the element face reaches the separation strain, all springs at this face are removed and elements are no longer connected until a collision occurs, at which point they collide together as rigid bodies.

Separation strain represents the strain at which adjacent elements are totally separated at the connecting face. This parameter is not available in the elastic material model. For concrete, all springs between the adjacent faces



The applied element method (AEM) is a numerical analysis used in predicting the continuum and discrete behavior of structures.

including reinforcement bar springs are cut. If the elements meet again, they will behave as two different rigid bodies that have now contacted each other. For steel, the bars are cut if the stress point reaches ultimate stress or if the concrete reaches the separation strain.

Automatic Element Contact/Collision

Contact or collision is detected without any user intervention. Elements are able to separate, contract and/or make contact with other elements. In AEM three contact methods include Corner-to-Face, Edge-to-Edge, and Corner-to-Ground.

2. STIFFNESS MATRIX

The spring stiffness in a 2D model can be calculated from the following equations:

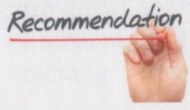
$$K_n = \frac{E \cdot T \cdot d}{a}$$

$$K_s = \frac{G \cdot T \cdot d}{a}$$

Where d is the distance between springs, T is the thickness of the element, a is the length of the representative area, E is the Young's modulus, and G is the shear modulus of the material. The above equation's indicate that each spring represents the stiffness of an area (T·d) within the length of the studied material.

To model reinforcement bars embedded in concrete, a spring is placed inside the element at the location of the bar; the area (T·d) is replaced by the actual cross section area of the reinforcement bar. Similar to modeling embedded steel sections, the area (T·d) may be replaced by the area of the steel section represented by the spring.

Although, the element motion moves as a rigid body, its internal deformations are represented by the spring deformation around each element. This means the element shape does not change during analysis, but the behavior of assembly of elements is deformable. The two elements are assumed to be connected by only one pair of normal and shear springs. To have a general stiffness matrix, the locations of element and contact springs are assumed in a general position. The stiffness matrix components corresponding to each



For concrete, all springs between the adjacent faces including reinforcement bar springs are cut.

degree of freedom are determined by assuming a unit displacement in the studied direction and by determining forces at the centroid of each element. The 2D element stiffness matrix size is 6 × 6; the components of the upper left quarter of the stiffness matrix are shown below:

$$\begin{bmatrix} \sin^2(\theta+\alpha)K_n + \cos^2(\theta+\alpha)K_s & -K_n\sin(\theta+\alpha)\cos(\theta+\alpha) + K_s\sin(\theta+\alpha)\cos(\theta+\alpha) & \cos(\theta+\alpha)K_sL\sin(\alpha) - \sin(\theta+\alpha)K_nL\cos(\alpha) \\ -K_n\sin(\theta+\alpha)\cos(\theta+\alpha) + K_s\sin(\theta+\alpha)\cos(\theta+\alpha) & \sin^2(\theta+\alpha)K_s + \cos^2(\theta+\alpha)K_n & \cos(\theta+\alpha)K_nL\cos(\alpha) + \sin(\theta+\alpha)K_sL\sin(\alpha) \\ \cos(\theta+\alpha)K_sL\sin(\alpha) - \sin(\theta+\alpha)K_nL\cos(\alpha) & \cos(\theta+\alpha)K_nL\cos(\alpha) + \sin(\theta+\alpha)K_sL\sin(\alpha) & L^2\cos^2(\alpha)K_n + L^2\sin^2(\alpha)K_s \end{bmatrix}$$

The stiffness matrix depends on the contact spring stiffness and the spring location. The stiffness matrix is for only one pair of contact springs. However, the global stiffness matrix is determined by summing up the stiffness matrices of individual pairs of springs around each element. Consequently, the developed stiffness matrix has total effects from all pairs of springs, according to the stress situation around the element. This technique can be used in both load and displacement control cases. The 3D stiffness matrix may be deduced similarly.

Applications

The applied element method is currently being used in the following applications:

- Structural vulnerability assessment
  - o Progressive collapse
  - o Blast analysis
  - o Impact analysis
- Seismic analysis
- Forensic engineering
- Performance based design

IMPORTANT

To have a general stiffness matrix, the locations of element and contact springs are assumed in a general position.

- Demolition analysis
- Glass performance analysis
- Visual effects

Generalized Finite Element Method

The Generalized Finite Element Method (GFEM) uses local spaces consisting of functions, not necessarily polynomials, that reflect the available information on the unknown solution and thus ensure good local approximation. Then a partition of unity is used to "bond" these spaces together to form the approximating subspace. The effectiveness of GFEM has been shown when applied to problems with domains having complicated boundaries, problems with micro-scales, and problems with boundary layers.

Mixed Finite Element Method

The mixed finite element method is a type of finite element method in which extra independent variables are introduced as nodal variables during the discretization of a partial differential equation problem.

Variable – Polynomial

The hp-FEM combines adaptively, elements with variable size h and polynomial degree p in order to achieve exceptionally fast, exponential convergence rates.

hpk-FEM

The hpk-FEM combines adaptively, elements with variable size h, polynomial degree of the local approximations p and global differentiability of the local approximations (k-1) to achieve best convergence rates.

XFEM

The Extended Finite Element Method (XFEM) is a numerical technique based on the generalized finite element method (GFEM) and the partition of unity method (PUM). It extends the classical finite element method by enriching the solution space for solutions to differential equations with discontinuous functions. Extended finite element methods enrich the approximation space so that it can naturally reproduce the challenging feature associated with the problem of interest: the discontinuity, singularity, boundary layer, etc. It was shown that for some problems, such an embedding of the problem's feature into the approximation space can significantly improve convergence rates and accuracy.

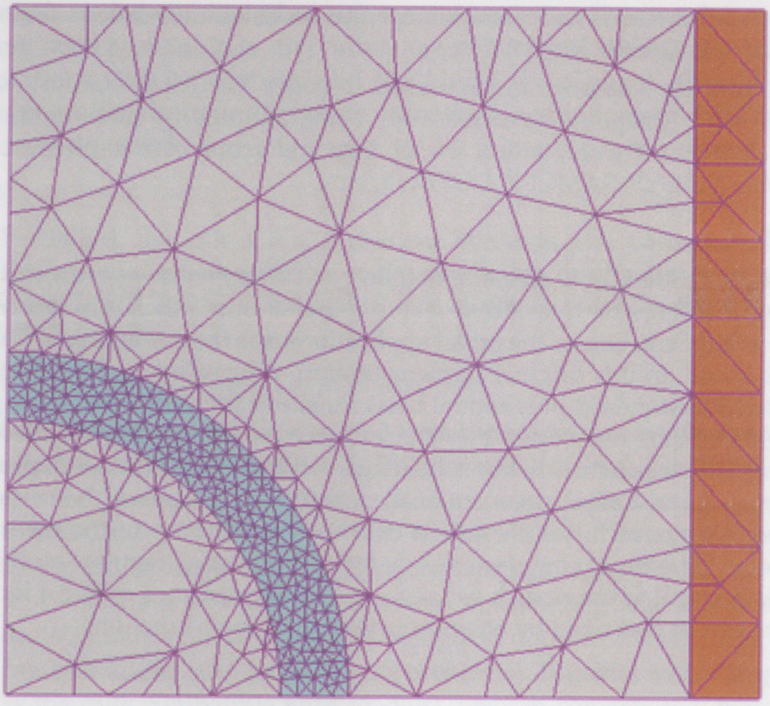


Fig. 11.1 Extended Finite Element Method

2D FEM mesh, the triangles are the elements, the vertices are the nodes. The finite element method (FEM) has been the tool of choice since civil engineer Ray W. Clough in 1940 derived the stiffness matrix of a 3-node triangular finite element (and coined the name). The precursors of FEM were elements built-up from bars (Hrennikoff, Argyris, Turner) and a conceptual variation approach suggested by R. Courant. Today, the FEM is used to model a much wider range of physical phenomena.

Moreover, treating problems with discontinuities with XFEMs suppresses the need to mesh and re-mesh the discontinuity surfaces, thus alleviating the computational costs and projection errors associated with conventional finite element methods, at the cost of restricting the discontinuities to mesh edges.

The Extended Finite Element Method (XFEM), is a numerical technique based on the generalized finite element method (GFEM) and the partition of unity method (PUM). It extends the classical finite element method (FEM) approach by enriching the solution space for solutions to differential equations with discontinuous functions.

Several research codes implement this technique to various degrees: 1. GetFEM++ 2. xfem++ 3. openxfem++

XFEM has also been implemented in codes like Altair Radios, ASTER, Morfeo, and Abaqus. It is increasingly being adopted by other commercial finite element software, with a few plugins and actual core implementations available (ANSYS, SAMCEF, OOFELIE, etc.).

Rationale

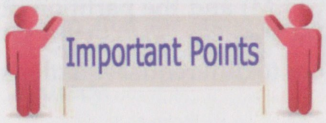
The extended finite element method was developed to ease difficulties in solving problems with localized features that are not efficiently resolved by mesh refinement. One of the initial applications was the modelling of fractures in a material. In this original implementation, discontinuous basis functions are added to standard polynomial basis functions for nodes that belonged to elements that are intersected by a crack to provide a basis that included crack opening displacements. A key advantage of XFEM is that in such problems the finite element mesh does not need to be updated to track the crack path. Subsequent research has illustrated the more general use of the method for problems involving singularities, material interfaces, regular meshing of microstructural features such as voids, and other problems where a localized feature can be described by an appropriate set of basis functions.

Principle

Enriched finite element methods extend, or enrich, the approximation space so that it is able to naturally reproduce the challenging feature associated with the problem of interest: the discontinuity, singularity, boundary layer, etc. It was shown that for some problems, such an embedding of the problem's feature into the approximation space can significantly improve convergence rates and accuracy. Moreover, treating problems with discontinuities with eXtended Finite Element Methods suppresses the need to mesh and remesh the discontinuity surfaces, thus alleviating the computational costs and projection errors associated with conventional finite element methods, at the cost of restricting the discontinuities to mesh edges.

Scaled Boundary Finite Element Method (SBFEM)

The introduction of the scaled boundary finite element method (SBFEM) came from Song and Wolf (1997). The SBFEM has been one of the most profitable contributions in the area of numerical analysis of fracture



It is a semi-analytical fundamental-solutionless method which combines the advantages of both the finite element formulations and procedures and the boundary element discretization.

mechanics problems. It is a semi-analytical fundamental-solutionless method which combines the advantages of both the finite element formulations and procedures and the boundary element discretization. However, unlike the boundary element method, no fundamental differential solution is required.

S-FEM

The S-FEM, Smoothed Finite Element Methods, is a particular class of numerical simulation algorithms for the simulation of physical phenomena. It was developed by combining meshfree methods with the finite element method.

History of Smoothed Finite Element Methods

The development of S-FEM started from the works on meshfree methods, where the so-called weakened weak (W2) formulation based on the G space theory[4] were developed. The W2 formulation offers possibilities for formulate various (uniformly) "soft" models that works well with triangular meshes. Because triangular mesh can be generated automatically, it becomes much easier in re-meshing and hence automation in modeling and simulation. In addition, W2 models can be made soft enough (in uniform fashion) to produce upper bound solutions (for force-driving problems). Together with stiff models (such as the fully compatible FEM models), one can conveniently bound the solution from both sides. This allows easy error estimation for generally complicated problems, as long as a triangular mesh can be generated.

Typical W2 models are the Smoothed Point Interpolation Methods (or S-PIM). The S-PIM can be node-based (known as NS-PIM or LC-PIM), edge-based (ES-PIM), and cell-based (CS-PIM). The NS-PIM was developed using the so-called SCNI technique. It was then discovered that NS-PIM is capable of producing upper bound solution and volumetric locking free. The ES-PIM is found superior in accuracy, and CS-PIM behaves in between the NS-PIM and ES-PIM. Moreover, W2 formulations allow the use of polynomial and radial basis functions in the creation of shape functions (it accommodates the discontinuous displacement functions, as long as it is in G1 space), which opens further rooms for future developments.

The S-FEM is largely the linear version of S-PIM, but with most of the properties of the S-PIM and much simpler. It has also variations of NS-FEM, ES-FEM and CS-FEM. The major property of S-PIM can be found also in S-FEM.

Description

The essential idea in the S-FEM is to use a finite element mesh (in particular triangular mesh) to construct numerical models of good performance. This is achieved by modifying the compatible strain field, or construct a strain field using only the displacements, hoping a Galerkin model using the modified/constructed strain field can deliver some good properties. Such a modification/construction can be performed within elements but more often beyond the elements (meshfree concepts): bring in the information from the neighboring elements.

Naturally, the strain field has to satisfy certain conditions, and the standard Galerkin weak form needs to be modified accordingly to ensure the stability and convergence. A comprehensive review of S-FEM covering both methodology and applications can be found in ("Smoothed Finite Element Methods (S-FEM): An Overview and Recent Developments").

Spectral Element Method

In the numerical solution of partial differential equations, a topic in mathematics, the spectral element method (SEM) is a formulation of the finite element method (FEM) that uses high degree piecewise polynomials as basis functions. The spectral element method was introduced in a 1984 paper[1] by A. T. Patera. Although Patera is credited with development of the method, his work was a rediscovery of an existing method.

Spectral element methods combine the geometric flexibility of finite elements and the acute accuracy of spectral methods. Spectral methods are the approximate solution of weak form partial equations that are based on high-order Lagrangian interpolants and used only with certain quadrature rules.[15]

Discussion

The spectral method expands the solution in trigonometric series, a chief advantage being that the resulting method is of very high order. This approach relies on the fact that trigonometric polynomials are an orthonormal basis for $L^2(\Omega)$. The spectral element method chooses instead a high degree piecewise polynomial basis functions, also achieving a very high order of accuracy. Such polynomials are usually orthogonal Chebyshev polynomials or very high order Legendre polynomials over non-uniformly spaced nodes. In SEM computational error decreases exponentially as the order of approximating polynomial, therefore a fast convergence of solution to the exact solution is realized with fewer degrees of freedom of the structure in comparison with FEM.

In structural health monitoring, FEM can be used for detecting large flaws in a structure, but as the size of the flaw is reduced there is a need to use a high frequency wave with a small wavelength. Therefore, the FEM mesh must be much finer, resulting in increased computational time and an inexact solution. SEM, with fewer degrees of freedom per node, can be useful for detecting small flaws. Non-uniformity of nodes helps to make the mass matrix diagonal, which saves time and memory and is also useful for adopting a central difference method (CDM). The disadvantages of SEM include difficulty in modeling complex geometry, compared to the flexibility of FEM.

Although the method can be applied with a modal piecewise orthogonal polynomial basis, it is most often implemented with a nodal tensor product Lagrange basis. The method gains its efficiency by placing the nodal points at the Legendre-Gauss-Lobatto (LGL) points and performing the Galerkin method integrations with a reduced Gauss-Lobatto quadrature using the same nodes. With this combination, simplifications result such that mass lumping occurs at all nodes and a collocation procedure results at interior points.

The most popular applications of the method are in computational fluid dynamics and modeling seismic wave propagation.

A-priori Error Estimate

The classic analysis of Galerkin methods and Céa's lemma holds here and it can be shown that, if u is the solution of the weak equation, u_N is the approximate solution and u $H^{s+1}()$:

$$\| u - u_N \|_{H^1(\Omega)} \leq C_s N^{-s} \| u \|_{H^{s+1}(\Omega)}$$

where C is independent from N and s is no larger than the degree of the piecewise polynomial basis. Similar results can be obtained to bound the error in stronger topologies. If k + 1

$$\| u - u_N \|_{H^k(\Omega)} \leq C_{s,k} N^{k-1-s} \| u \|_{H^{s+1}(\Omega)}$$

As we increase N, we can also increase the degree of the basis functions. In this case, if u is an analytic function:

$$\| u - u_N \|_{H^1(\Omega)} \leq C \exp(-\gamma N)$$

where γ depends only on u.

The Hybrid-Collocation-Galerkin possesses some superconvergence properties. The LGL form of SEM is equivalent, so it achieves the same superconvergence properties.

Related Methods

- G-NI or SEM-NI are the most used spectral methods. The Galerkin formulation of spectral methods or spectral element methods, for G-NI or SEM-NI respectively, is modified and Gauss-Lobatto integration is used instead of integrals in the definition of the bilinear form a and in the functional F. Their convergence is a consequence of Strang's lemma.
- SEM is a Galerkin based FEM (finite element method) with Lagrange basis (shape) functions and reduced numerical integration by Lobatto quadrature using the same nodes.
- The pseudospectral method, orthogonal collocation, differential quadrature method, and G-NI are different names for the same method. These methods employ global rather than piecewise polynomial basis functions. The extension to a piecewise FEM or SEM basis is almost trivial.
- The spectral element method uses a tensor product space spanned by nodal basis functions associated with Gauss–Lobatto points. In contrast, the p-version finite element method spans a space of high order polynomials by nodeless basis functions, chosen approximately orthogonal for numerical stability. Since not all interior basis functions need to be present, the p-version finite element method can create a space that contains all polynomials up to a given degree with fewer degrees of freedom. However, some speedup techniques possible in spectral methods due to their tensor-product character are no longer available. The name p-version means that accuracy is increased by increasing the order of the approximating polynomials (thus, p) rather than decreasing the mesh size, h.
- The hp finite element method (hp-FEM) combines the advantages of the h and p refinements to obtain exponential convergence rates.

Meshfree Methods

In the field of numerical analysis, meshfree methods are those that do not require connection between nodes of the simulation domain, i.e. a mesh, but are rather based on interaction of each node with all its neighbors. As a consequence, original extensive properties such as mass or kinetic energy are no longer assigned to mesh elements but rather to the single nodes. Meshfree methods enable the simulation of some otherwise difficult types of problems, at the cost of extra computing time and programming effort. The absence of a mesh allows Lagrangian simulations, in which the nodes can move according to the velocity field.

Motivation

Numerical methods such as the finite difference method, finite-volume method, and finite element method were originally defined on meshes of data points. In such a mesh, each point has a fixed number of predefined neighbors, and this connectivity between neighbors can be used to define mathematical operators like the derivative. These operators are then used to construct the equations to simulate—such as the Euler equations or the Navier–Stokes equations.

But in simulations where the material being simulated can move around (as in computational fluid dynamics) or where large deformations of the material can occur (as in simulations of plastic materials), the connectivity of the mesh can be difficult to maintain without introducing error into the simulation. If the mesh becomes tangled or degenerate during simulation, the operators defined on it may no longer give correct values. The mesh may be recreated during simulation (a process called remeshing), but this can also introduce error, since all the existing data points must be mapped onto a new and different set of data points. Meshfree methods are intended to remedy these problems. Meshfree methods are also useful for:

- Simulations where creating a useful mesh from the geometry of a complex 3D object may be especially difficult or require human assistance
- Simulations where nodes may be created or destroyed, such as in cracking simulations
- Simulations where the problem geometry may move out of alignment with a fixed mesh, such as in bending simulations
- Simulations containing nonlinear material behavior, discontinuities or singularities

Recent Development

The primary areas of advancement in meshfree methods are to address issues with essential boundary enforcement, numerical quadrature, and contact and large deformations.The common weak form requires strong enforcement of the essential boundary conditions, yet meshfree methods in general lack the Kronecker delta property. This make essential boundary condition enforcement non-trivial, at least more difficult than the Finite element method, where they can be imposed directly. Techniques have been developed to overcome this difficulty and impose conditions strongly. Several methods have been developed to impose the essential boundary conditions weakly, including Lagrange multipliers, Nitche's method, and the penalty method.

As for quadrature, nodal integration is generally preferred which offers simplicity, efficiency, and keeps the meshfree method free of any mesh (as opposed to using Gauss quadrature, which necessitates a mesh to generate quadrature points and weights). Nodal integration however, suffers from numerical instability due to underestimation of strain energy associated with short-wavelength modes, and also yields inaccurate and non-convergent results due to under-integration of the weak form. One major advance in numerical integration has been the development of a stabilized conforming nodal integration (SCNI) which provides a nodal integration method which does not suffer from either of these problems. The method is based on strain-smoothing which satisfies the first order patch test.

However, it was later realized that low-energy modes were still present in SCNI, and additional stabilization methods have been developed. This method has been applied to a variety of problems including thin and thick plates, poromechanics, convection-dominated problems, among others.[21] More recently, a framework has been developed to pass arbitrary-order patch tests, based on a Petrov–Galerkin method.

One recent advance in meshfree methods aims at the development of computational tools for automation in modeling and simulations. This is enabled by the so-called weakened weak (W2) formulation based on the G space theory. The W2 formulation offers possibilities to formulate various (uniformly) "soft" models that work well with triangular meshes. Because a triangular mesh can be generated automatically, it becomes much easier in re-meshing and hence enables automation in modeling and simulation. In addition, W2 models can be made soft enough (in uniform fashion) to produce upper bound solutions (for force-driving problems). Together with stiff models (such as the fully compatible FEM models), one can conveniently bound the solution from both sides. This allows easy error estimation for generally complicated problems, as long as a triangular mesh can be generated. Typical W2 models are the Smoothed Point Interpolation Methods (or S-PIM). The S-PIM can be node-based (known as NS-PIM or LC-PIM), edge-based (ES-PIM), and cell-based (CS-PIM).

The NS-PIM was developed using the so-called SCNI technique. It was then discovered that NS-PIM is capable of producing upper bound solution and volumetric locking free. The ES-PIM is found superior in accuracy, and CS-PIM behaves in between the NS-PIM and ES-PIM. Moreover, W2 formulations allow the use of polynomial and radial basis functions in the creation of shape functions (it accommodates the discontinuous displacement functions, as long

as it is in G1 space), which opens further rooms for future developments. The W2 formulation has also led to the development of combination of meshfree techniques with the well-developed FEM techniques, and one can now use triangular mesh with excellent accuracy and desired softness. A typical such a formulation is the so-called smoothed finite element method (or S-FEM). The S-FEM is the linear version of S-PIM, but with most of the properties of the S-PIM and much simpler.

It is a general perception that meshfree methods are much more expensive than the FEM counterparts. The recent study has found however, some meshfree methods such as the S-PIM and S-FEM can be much faster than the FEM counterparts.

The S-PIM and S-FEM works well for solid mechanics problems. For CFD problems, the formulation can be simpler, via strong formulation. A Gradient Smoothing Methods (GSM) has also been developed recently for CFD problems, implementing the gradient smoothing idea in strong form. The GSM is similar to [FVM], but uses gradient smoothing operations exclusively in nested fashions, and is a general numerical method for PDEs. Nodal integration has been proposed as a technique to use finite elements to emulate a meshfree behaviour. However, the obstacle that must be overcome in using nodally integrated elements is that the quantities at nodal points are not continuous, and the nodes are shared among multiple elements.

Finite Element Limit Analysis

A finite element limit analysis (FELA) uses optimisation techniques to directly compute the upper or lower bound plastic collapse load (or limit load) for a mechanical system rather than time stepping to a collapse load, as might be undertaken with conventional non-linear finite element techniques. The problem may be formulated in either a kinematic or equilibrium form.

The technique has been used most significantly in the field of soil mechanics for the determination of collapse loads for geotechnical problems (e.g. slope stability analysis). An alternative technique which may be used to undertake similar direct plastic collapse computations using optimization is Discontinuity layout optimization.

Stretched Grid Method

The stretched grid method (SGM) is a numerical technique for finding approximate solutions of various mathematical and engineering problems that can be related to an elastic grid behavior. In particular, meteorologists

use the stretched grid method for weather prediction[1] and engineers use the stretched grid method to design tents and other tensile structures.

FEM and BEM Mesh Refinement

In recent decades the finite element and boundary element methods (FEM and BEM) have become a mainstay for industrial engineering design and analysis. Increasingly larger and more complex designs are being simulated using the FEM or BEM. However, some problems of FEM and BEM engineering analysis are still on the cutting edge. The first problem is a reliability of engineering analysis that strongly depends upon the quality of initial data generated at the pre-processing stage. It is known that automatic element mesh generation techniques at this stage have become commonly used tools for the analysis of complex real-world models.[2] With FEM and BEM increasing in popularity comes the incentive to improve automatic meshing algorithms. However, all of these algorithms can create distorted and even unusable grid elements. Several techniques exist which can take an existing mesh and improve its quality. For instance smoothing (also referred to as mesh refinement) is one such method, which repositions nodal locations, so as to minimize element distortion. The Stretched Grid Method (SGM) allows the obtaining of pseudo-regular meshes very easily and quickly in a one-step solution(see).

Let one assume that there is an arbitrary triangle grid embedded into plane polygonal single-coherent contour and produced by an automeshing procedure. It may be assumed further that the grid considered as a physical nodal system is distorted by a number of distortions. It is supposed that the total potential energy of this system is proportional to the length of some n -dimensional vector with all network segments as its components.

Loubignac Iteration

Loubignac iteration is an iterative method in finite element methods.

Crystal Plasticity Finite Element Method (CPFEM)

Crystal plasticity finite element method (CPFEM) is an advanced numerical tool developed by Franz Roters. Metals can be regarded as crystal aggregates and it behave anisotropy under deformation, for example, abnormal stress and strain localization. CPFEM based on slip (shear strain rate) can calculate dislocation, crystal orientation and other texture information to consider crystal anisotropy during the routine. Now it has been applied in the numerical study of material deformation, surface roughness, fractures and so on.

Link with the Gradient Discretization Method

Some types of finite element methods (conforming, nonconforming, mixed finite element methods) are particular cases of the gradient discretization method (GDM). Hence the convergence properties of the GDM, which are established for a series of problems (linear and non-linear elliptic problems, linear, nonlinear, and degenerate parabolic problems), hold as well for these particular finite element methods.

Comparison to the Finite Difference Method

The finite difference method (FDM) is an alternative way of approximating solutions of PDEs. The differences between FEM and FDM are:

- The most attractive feature of the FEM is its ability to handle complicated geometries (and boundaries) with relative ease. While FDM in its basic form is restricted to handle rectangular shapes and simple alterations thereof, the handling of geometries in FEM is theoretically straightforward.
- FDM is not usually used for irregular CAD geometries but more often rectangular or block shaped models.
- The most attractive feature of finite differences is that it is very easy to implement.
- There are several ways one could consider the FDM a special case of the FEM approach. E.g., first-order FEM is identical to FDM for Poisson's equation, if the problem is discretized by a regular rectangular mesh with each rectangle divided into two triangles.
- There are reasons to consider the mathematical foundation of the finite element approximation more sound, for instance, because the quality of the approximation between grid points is poor in FDM.
- The quality of a FEM approximation is often higher than in the corresponding FDM approach, but this is extremely problem-dependent and several examples to the contrary can be provided.

Generally, FEM is the method of choice in all types of analysis in structural mechanics (i.e. solving for deformation and stresses in solid bodies or dynamics of structures) while computational fluid dynamics (CFD) tend to use FDM or other methods like finite volume method (FVM). CFD problems usually require discretization of the problem into a large number of cells/gridpoints (millions and more), therefore the cost of the solution favors simpler, lower-order approximation within each cell. This is especially true for 'external flow' problems, like airflow around the car or airplane, or weather simulation.

Application

A variety of specializations under the umbrella of the mechanical engineering discipline (such as aeronautical, biomechanical, and automotive industries) commonly use integrated FEM in the design and development of their products. Several modern FEM packages include specific components such as thermal, electromagnetic, fluid, and structural working environments. In a structural simulation, FEM helps tremendously in producing stiffness and strength visualizations and also in minimizing weight, materials, and costs.

FEM allows detailed visualization of where structures bend or twist, and indicates the distribution of stresses and displacements. FEM software provides a wide range of simulation options for controlling the complexity of both modeling and analysis of a system. Similarly, the desired level of accuracy required and associated computational time requirements can be managed simultaneously to address most engineering applications. FEM allows entire designs to be constructed, refined, and optimized before the design is manufactured. The mesh is an integral part of the model and it must be controlled carefully to give the best results. Generally the higher the number of elements in a mesh, the more accurate the solution of the discretized problem. However, there is a value at which the results converge and further mesh refinement does not increase accuracy.

This powerful design tool has significantly improved both the standard of engineering designs and the methodology of the design process in many industrial applications. The introduction of FEM has substantially decreased the time to take products from concept to the production line. It is primarily through improved initial prototype designs using FEM that testing and development have been accelerated. In summary, benefits of FEM include increased accuracy, enhanced design and better insight into critical design parameters, virtual prototyping, fewer hardware prototypes, a faster and less expensive design cycle, increased productivity, and increased revenue.

In the 1990s FEA was proposed for use in stochastic modelling for numerically solving probability models and later for reliability assessment.

CONCLUSION

The finite element method started with significant promise in the modeling of several mechanical applications related to aerospace and civil engineering. The applications of the finite element method are only now starting to reach their potential. One of the most exciting prospects is its application in coupled problems such as fluid-structure interaction, thermomechanical,

thermochemical, thermo-chemo-mechanical problems, biomechanics, biomedical engineering, piezoelectric, ferroelectric, and electromagnetics.

FEA was developed originally for numerical solutions of complex problems in solid mechanics. FEA is by far the most widely used and versatile technique for simulating deformable solids. This chapter gives an overview of the physical and mathematic background required to understand the FEA implementation for solid mechanics' problems. The physical behaviors of mechanical structures or systems are analyzed, and the minimum potential energy principle is used to develop element models. The procedures for FEA modeling are discussed for a few of classic solid mechanics' problems such as truss structure, plane stress, plane strain, modal analysis, as well as fatigue analysis.

FEA-based simulations are used to serve different design purposes. Two basic design activities are design analysis and design synthesis. In design analysis, all the design variables and system parameters are given; an FEA model is to find the system state or response to external loads. It is a type of task for design analysis, which is commonly referred as what–if simulation. In design synthesis, some design variables are to be determined based on the specified design criteria. In other words, design variables are given as changeable ranges rather than values, FEA models are applied to evaluate the responses of a system; although design variables can vary in a design space.

REFERENCES

- Chen, J. S., Wu, C. T., Yoon, S. and You, Y. (2001). A stabilized conforming nodal integration for Galerkin mesh-free methods. Int. J. Numer. Meth. Eng. 50: 435–466.
- Dai KY, Liu GR (2007) Free and forced vibration analysis using the smoothed finite element method (SFEM). Journal of Sound and Vibration; 301: 803-820.
- Dai KY, Liu GR, Nguyen-Thoi T (2007) An n-sided polygonal smoothed finite element method (nSFEM) for solid mechanics. Finite Elements in Analysis and Design; 43: 847-860.
- G. R. Liu and G. Y. Zhang. Upper bound solution to elasticity problems: A unique property of the linearly conforming point interpolation method (LC-PIM). International Journal for Numerical Methods in Engineering, 74: 1128-1161, 2008.
- HATEM, Tagel-Din; Kimiro MEGURO, K (August 1–6, 2004). Dynamic Modeling of Dip-Slip Faults for Studying Ground Surface Deformation Using Applied Element Method. Vancouver, Canada: Proceedings of the 13th World Conference on Earthquake Engineering.
- Jiang C, Zhang ZQ, Liu GR, Han X, Zeng W, An edge-based/node-based selective smoothed finite element method using tetrahedrons for cardiovascular tissues, Engineering Analysis with Boundary Elements Vol.59, 62-77, 2015.

- Li Y, Liu GR, Zhang GY, An adaptive NS/ES-FEM approach for 2D contact problems using triangular elements, Finite Elements in Analysis and Design Vol.47 Issue: 3, 256-275, 2011.
- Liu GR, Dai KY, Nguyen-Thoi T (2007) A smoothed finite element method for mechanics problems. Computational Mechanics; 39: 859-877.
- Liu GR, Nguyen-Thoi T, Lam KY (2009) A novel FEM by scaling the gradient of strains with factor α (αFEM). Computational Mechanics; 43: 369-391.
- Liu GR, Nguyen-Thoi T, Lam KY (2009) An edge-based smoothed finite element method (ES-FEM) for static, free and forced vibration analyses in solids. Journal of Sound and Vibration; 320: 1100-1130.
- Liu GR, Nguyen-Thoi T, Nguyen-Xuan H, Lam KY (2009) A node-based smoothed finite element method (NS-FEM) for upper bound solutions to solid mechanics problems. Computers and Structures; 87: 14-26.
- Liu GR, Nguyen-Xuan H, Nguyen-Thoi T, Xu X (2009) A novel weak form and a superconvergent alpha finite element method (SαFEM) for mechanics problems using triangular meshes. Journal of Computational Physics; 228: 4055-4087.
- Liu GR, Zhang GY, Dai KY, Wang YY, Zhong ZH, Li GY and Han X, A linearly conforming point interpolation method (LC-PIM) for 2D solid mechanics problems, International Journal of Computational Methods, 2(4): 645-665, 2005.
- Liu, G.R. 2nd edn: 2009 Mesh Free Methods, CRC Press. 978-1-4200-8209-9.
- Liu, G.R., 2010 Smoothed Finite Element Methods, CRC Press, Lysmer, J. (1970). Limit analysis of plane problems in soil mechanics. Journal of the Soil Mechanics and Foundations Division ASCE 96(4), 1311–1334.
- Mayorka, Paola; Kimiro Meguro, K (2005). Blast Testing and Research Bridge at the Tenza Viaduct. Japan: University of Missouri-Rolla, TSWG Contract Number N4175-05-R-4828, Final Report of Task 1.
- Mayorka, Paola; Kimiro Meguro, K (October 2003). "Modeling Masonry Structures using the Applied Element Method". Seisan Kenkyu. Japan: Institute of Industrial Science, The University of Tokyo. 55 (6): 123–126. ISSN 1881-2058. Retrieved 2009-08-10.
- Meguro, K.; Tagel-Din, H. (2000). "Applied element method for structural analysis: Theory and application for linear materials". Structural Engineering/ Earthquake Engineering. Japan: Japan Society of Civil Engineers(JSCE). 17 (1): 21–35. F0028A. Archived from the original on 2012-02-29. Retrieved 2009-08-10.
- Nguyen-Thoi T, Liu GR, Lam KY, GY Zhang (2009) A Face-based Smoothed Finite Element Method (FS-FEM) for 3D linear and nonlinear solid mechanics problems using 4-node tetrahedral elements. International Journal for Numerical Methods in Engineering; 78: 324-353.
- Tagel-Din, H.; Meguro, K (2000). "Applied Element Method for Simulation of Nonlinear Materials: Theory and Application for RC Structures". Structural Engineering/Earthquake Engineering. Japan: Japan Society of Civil Engineers(JSCE). 17 (2): 137–148. Retrieved 2009-08-10.

- Tagel-Din, H.; Meguro, K (2002). "AEM Used for Large Displacement Structure Analysis" (PDF). Journal of Natural Disaster Science. Japan. 24 (1): 25–34. Retrieved 2009-08-10.[permanent dead link].
- Tagel-Din, H.; Meguro, Kimiro (November 2001). "Applied Element Simulation of RC Structures under Cyclic Loading". Journal of Structural Engineering. Japan: ASCE. 127 (11): 137–148. doi:10.1061/(ASCE)0733-9445(2001)127:11(1295). ISSN 0733-9445. Retrieved 2009-08-10.
- Tagel-Din, Hatem; Kimiro Meguro, K (January 30 – February 4, 2000). Analysis of a Small Scale RC Building Subjected to Shaking Table Tests using Applied Element Method. New Zealand: Proceedings of the 12th World Conference on Earthquake Engineering. pp. 25–34.
- W. Zeng, G.R. Liu. Smoothed finite element methods (S-FEM): An overview and recent developments. Archives of Computational Methods in Engineering, 2016, doi: 10.1007/s11831-016-9202-3.
- Zeng W, Liu GR, Li D, Dong XW (2016) A smoothing technique based beta finite element method (βFEM) for crystal plasticity modeling. Computers and Structures; 162: 48-67.
- Zhang G.R. Liu, G.R. A normed G space and weakened weak (W2) formulation of a cell-based Smoothed Point Interpolation Method. International Journal of Computational Methods, 6(1): 147-179, 2009.
- Zhang G.R. Liu, G.R. Edge-based Smoothed Point Interpolation Methods. International Journal of Computational Methods, 5(4): 621-646, 2008.

MCQs

1. **The finite element method formulation of the problem results in a system of**
   a. algebraic equations
   b. logical equations
   c. Arthimatic equations
   d. flow equations
2. **FEM gives accurate representation of**
   a. real geometry
   b. complex geometry
   c. real and complex geometry
   d. constant geometry
3. **Finite element method is also called**
   a. infinite element analysis
   b. frequency element analysis
   c. finite element analysis
   d. partial element analysis

4. **Numerical algorithms are based on**
   a. FEM and FDTD
   b. FEM and IFEM
   c. TD and FD
   d. FEM and FD
5. **To solve the FEM problem, it subdivides a large problem into smaller, simpler parts that are called**
   a. finite elements
   b. infinite elements
   c. dynamic elements
   d. static elements

ANSWERS

1 (a) 2 (b) 3 (c) 4 (a) 5 (a)

Index

A

Adaptive finite element methods (AFEMs) 74
aerodynamic 111
algebraic 8, 51, 53, 65, 66, 70, 84, 127, 174, 175, 188, 198, 204, 206, 233
alternating current (AC) 107
analogous 33, 37, 175
Analytical 12, 13, 36, 89
aneurysm 140
anisotropic 12, 102, 103, 104
aperiodic 174
asymptotic 32, 101, 174, 193, 198
Axisymmetric 157, 161, 162, 179, 180

B

Biharmonic 23, 40, 47, 122, 128, 136, 155, 157, 165
Boundary conditions (B.C.) 64
boundary element method (BEM) 122, 126, 188, 192
boundary value problem (BVP) 55, 116

C

computer-aided design (CAD) 86

D

decentralization 89, 104
degrees of freedom (DOF) 148
direct current (DC) 107
discontinuous Galerkin finite element method (DGFEM) 35
Discrete element method, or (DEM) 214

E

eigenvalues 8, 33, 34, 65, 124, 126
eignenfunctions 8
Element-Free Galerkin Method (EFGM) 35
Elliptic 32, 34, 39, 181, 193, 212
Entity-Relationship Modeling (ER Modeling) 5
equilibrium 9, 13, 15, 51, 52, 57, 59, 82, 116, 129, 134, 139, 140, 160, 168, 169, 210, 227
equivalent 23, 55, 59, 140, 160, 187, 207, 223
Euler–Tricomi 32, 33
extended finite element method (XFEM) 35

F

finite difference methods (FDM) 35
finite element analysis (FEA) 35, 51, 53, 86
finite element method (FEM) 35, 51, 157, 169, 188, 219, 222
Finite volume 36
finite volume methods (FVM 35

G

gastrointestinal 140
generalized finite element method (GFEM) 35, 218, 219

H

harmonic 41, 42, 43, 44, 45, 51, 68, 78, 79, 80, 107, 119, 120, 121, 177, 199
hexahedron 92, 93
Hilbert 46, 56, 57, 154, 174, 199, 200, 201
holomorphic 43, 44
homogeneous 32, 33, 35, 67, 68, 69, 126, 142, 171, 172, 181, 198
homotopy 39, 198
hybrids 32
hyperbolic 31, 32, 33, 34, 35, 39, 40, 45, 83, 127, 177, 184

I

Integrated development environments (IDEs) 4
Interpolating Element-Free Galerkin Method (IEFGM) 35
isogeometric 54, 98
isomorphism 208
Isoparametric Elements 157, 169

K

Kinematic 24
Korteweg 32

L

Laplacian 40, 44, 190, 191
Left Hand Side (L.H.S.) 70
longitudinal 21, 115, 118, 140, 213
Lyapunov 39, 175, 184, 185

M

method of fundamental solution (MFS) 191
Methodologies 5

N

Neumann 43, 45, 62, 66, 67, 68, 69, 122
nonlinearities 32, 62, 176, 177

O

Object-Oriented Analysis and Design (OOAD and Model-Driven Architecture (MDA) 5
orthogonality 183, 187
oscillator 51, 177

P

parabolic 30, 32, 33, 39, 40, 184, 229
partial differential equations (PDE) 35, 53
Pendula 176
periodic 37, 174
perturbation 38, 39, 198
polyhedra 97, 207
polynomial 33, 53, 86, 91, 98, 99, 101, 122, 166, 173, 174, 218, 220, 221, 222, 223, 224, 226
posteriori 45, 54, 73, 74, 185, 186
preliminary 1, 14, 74, 100, 236
prismatic 13, 18, 116, 117, 149, 152
Pyramid 93
Python 3

Q

quadratic 33, 45, 98, 124, 163, 167, 181, 184
quasilinear 32

R

representative volume element (RVE) 141
Right Hand Side (R.H.S) 70
Robustness 2

S

Semianalytical 39
semidiscrete 60, 61
sinusoidal 36, 51, 107, 121
skewness 95, 96
spectral finite element method (SFEM) 35
statistical volume element (SVE) 141
stochastic finite elements (SFE) 141
symmetrical 52, 58, 127, 211

T

tetrahedrons 94, 231
thermodynamics 24, 26, 81, 112
thermomechanical 137, 230

U

ulterior 59
Ultrahyperbolic 34
Unified Modeling Language (UML) 5

V

Viscous 107, 108

FINITE ELEMENTS IN ENGINEERING

To fully comprehend and quantify any physical phenomenon, such as structural or fluid behaviour, thermal transfer, wave propagation, and biological cell growth, mathematics is required. Partial differential equations are used to describe the majority of these processes (PDEs). Numerical algorithms have been developed over the last few decades to allow a computer to solve these PDEs, and one of the most popular today is the finite element method. In two or three space variables, the FEM is a general numerical approach for solving partial differential equations (i.e., some boundary value problems). The FEM breaks a big system into smaller, simpler sections called finite elements in order to solve a problem. The classic domains of structural analysis, heat transfer, fluid flow, mass transport, and electromagnetic potential are typical issue areas of interest.

This book provides comprehensive insights into the field of finite elements. It talks extensively about the finite element analysis of harmonic problems and their other applications and uses. It deals with structural analysis, continuum mechanics problems, finite element meshes and biharmonic problems. It also focuses on harmonic problems, elliptic boundary value problems and variational formulation. It is compiled in such a manner, that it will provide in-depth knowledge about the theory and practice of the subject. Coherent flow of topics, student-friendly Language and extensive use of example make this book an invaluable source of knowledge. For all those who are interested in this topic, this book can prove to be an essential guide.

Saundra Peltier is a Ph.D. holder in Fluid Mechanics & Engineering from Public Research University, La Pampa. She has served as a Guest Lecturer for various universities across America and Europe. Her areas of expertise cover multidisciplinary areas: from design and optimisation of energy systems to fundamental problems focused on modelling and simulation. Peltier is author of more than 100 scientific papers and member of editorial board and scientific committee of a number of journals and conferences. She has been also a principal investigator in several projects for universities and companies.

M.Hill Didactics Co.

M.HILL DIDACTICS CO.
Email: enquiry@mhilldidactics.com
Website: mhilldidactics.com

ISBN : 978-9-35327-592-1