

THE UNIVERSITY OF CALGARY

Planar Dynamics of Topologically Variable
Multi Body Systems

by

Pawel Wierzba

A THESIS

SUBMITTED TO THE FACULTY OF GRADUATE STUDIES
IN PARTIAL FULFILMENT OF THE REQUIREMENTS FOR THE
DEGREE OF DOCTOR OF PHILOSOPHY

DEPARTMENT OF MECHANICAL ENGINEERING

CALGARY, ALBERTA

NOVEMBER 1990

© Pawel Wierzba 1990



National Library
of Canada

Bibliothèque nationale
du Canada

Canadian Theses Service Service des thèses canadiennes

Ottawa, Canada
K1A 0N4

The author has granted an irrevocable non-exclusive licence allowing the National Library of Canada to reproduce, loan, distribute or sell copies of his/her thesis by any means and in any form or format, making this thesis available to interested persons.

The author retains ownership of the copyright in his/her thesis. Neither the thesis nor substantial extracts from it may be printed or otherwise reproduced without his/her permission.

L'auteur a accordé une licence irrévocable et non exclusive permettant à la Bibliothèque nationale du Canada de reproduire, prêter, distribuer ou vendre des copies de sa thèse de quelque manière et sous quelque forme que ce soit pour mettre des exemplaires de cette thèse à la disposition des personnes intéressées.

L'auteur conserve la propriété du droit d'auteur qui protège sa thèse. Ni la thèse ni des extraits substantiels de celle-ci ne doivent être imprimés ou autrement reproduits sans son autorisation.

ISBN 0-315-66902-0

Canada

THE UNIVERSITY OF CALGARY
FACULTY OF GRADUATE STUDIES

The undersigned certify that they have read and recommended to the faculty of Graduate Studies for acceptance, a thesis entitled "Planar Dynamics of Topologically Variable Multi Body Systems" submitted by Pawel Wierzba in partial fulfilment of the requirements for the degree of Doctor in Philosophy in Mechanical Engineering.

Chairman, Dr. O. G. Vinogradov
Department of Mechanical Engineering

Dr. E. C. Mikulcik
Department of Mechanical Engineering

Dr. J. A. R. Blais
Department of Surveying Engineering

Dr. P. G. Glockner
Department of Mechanical Engineering

Dr. G.R. Pilkington, External Examiner
CANATEC Consultants Ltd., Calgary

November 1990

ABSTRACT

A general simulation methodology for analyzing two-dimensional (planar) multi body systems with variable topology has been proposed. A computer model capable of analyzing the systems with one-sided constraints has been created. The rigid bodies are approximated by disks or rigid conglomerates of disks, thus providing an ease in the determination of the system's topology. Line segments are used to define the system boundaries (wall) and the inter-boundary obstructions, which allows for creation (or approximation) of even very complex boundaries.

The potential flow theory is used to determine fluid flow velocity field within the system. The panel method is used for this purpose, which is directly compatible with line segment boundary description.

The rigid bodies present in the system, can be either predefined or generated. The rigid bodies can be generated at random with respect to their size, shape, position, and the frequency of appearances. Any type of probability density function for any of the above variables associated with the rigid bodies can be randomly sampled. Henceforth, a wide array of stochastic processes involving multi body systems can be modeled.

The mathematical description of the system's behaviour is

based on the Lagrangian dynamics. The Lagrangian equations of the second type together with a set of constraint equations are used to describe the motion of the system. The topological analysis is used to reduce the number of the differential equations of motion and the number of algebraic constraint equations.

The minimization of the number of the motion and constraint equations necessitated that one-sided constraints are treated as double-sided constraints until the corresponding contact (constraint) forces become tensile, at which point they are broken. It is for this purpose that an alternate (not based on Lagrangian approach) method of contact force computation has been developed. The method is based on both the dynamic and static (D'Alembert principle) analysis. This method also allows for explicit treatment of friction among the system's elements.

Collisions among the rigid bodies are detected and handled, and the corresponding impact loads calculated. Classical (Newtonian) theory of collisions is used.

The appropriate computer routines were programmed and the number of numerical experiments, some of which are presented in this report, were performed.

ACKNOWLEDGEMENTS

I would like to take this opportunity to express my sincere gratitude to my supervisor Dr. Oleg Vinogradov for his help, guidance, and encouragement during my work on the project.

I would like to acknowledge my indebtedness to the University of Calgary and National Research Council for providing financial support to me during my postgraduate term.

I would also like to extend special thanks to my mother Ida, and my brother Andrew, for support, encouragement, and assistance that they offered me during often difficult times.

Finally, I wish to express my gratefulness to my dear friends and colleagues, Arthur Springer and Per Nese, for their help, valuable suggestions, and constant exchange of ideas. Thank you guys.

Big thanks are also due to Nick Vogt for help with Auto Cad in the preparation of charts, to Arthur Moehrle for the encouragement, and to Maria Berry for taking care of all the administrative affairs.

TABLE OF CONTENTS

ABSTRACT	iii
ACKNOWLEDGEMENTS	v
TABLE OF CONTENTS	vi
LIST OF FIGURES	x
NOMENCLATURE	xiii

CHAPTER 1 INTRODUCTION AND OBJECTIVES

1.1 PROLOGUE	1
1.2 GENERAL	5
1.3 OBJECTIVES	10

CHAPTER 2 MODELLING OF THE MOVEMENT OF MULTI-RIGID BODY SYSTEMS : SUBJECT REVIEW

2.1 INTRODUCTION	12
2.2 TECHNIQUES OF MODELLING MULTI-RIGID SYSTEMS: A REVIEW OF VARIOUS EXISTING MODELS	15
2.3 PROPOSED TECHNIQUE OF MODELLING	27
2.3.1 General Approach	29
2.3.2 Topological Analysis	33
2.3.3 Events	45
2.4 CONCLUSIONS	54

CHAPTER 3 MATHEMATICAL MODEL

3.1	INTRODUCTION	56
3.2	LAGRANGIAN DYNAMICS	58
3.3	CONSTRAINT EQUATIONS	66
3.4	TREATMENT OF RIGID BODIES	70
3.5	PROPOSED SYSTEM OF GENERALIZED COORDINATES	74
3.6	CONCLUSIONS	78

CHAPTER 4 GENERATION OF RIGID BODIES

4.1	INTRODUCTION	79
4.2	RANDOM NUMBER GENERATORS	85
4.3	SAMPLING TECHNIQUES	90
4.4	METHODOLOGY OF GENERATING RIGID BODIES	104
4.4.1	Sampling for Size of Rigid Body	107
4.4.2	Sampling for Position	110
4.4.3	Sampling for Time interval	112
4.4.4	Positioning of Generated Rigid Bodies	117
4.5	SURFACE DENSITY CONCEPT	124
4.6	OTHER RELATED TOPICS	128
4.6.1	Other Shapes of Rigid Bodies	128
4.6.2	Generation of Random Shapes	132
4.7	CONCLUSIONS	144

CHAPTER 5 FLOW AROUND OBSTRUCTION

5.1	INTRODUCTION	145
-----	--------------	-----

5.2	EQUATIONS OF MOTION	147
5.3	POTENTIAL FLOW THEORY	151
5.4	PROPOSED METHOD OF FLOW COMPUTATION	157
5.5	VERIFICATION OF THE PROPOSED METHOD OF FLOW COMPUTATION	168
5.6	CONCLUSIONS	172
 CHAPTER 6 FORCES ON RIGID BODIES		
6.1	INTRODUCTION	174
6.2	EXTERNAL FORCES	175
6.3	INTERNAL FORCES	178
	6.3.1 Frictional Forces	179
	6.3.2 Impact Forces	186
6.4	CONCLUSIONS	190
 CHAPTER 7 CONTACT FORCES AMONG THE RIGID BODIES AND OBSTRUCTIONS		
7.1	INTRODUCTION	191
7.2	DYNAMIC EQUILIBRIUM ANALYSIS	195
	7.2.1 Disks with One Point of Contact	197
	7.2.2 Disks with Two Points of Contact	199
7.3	STATIC EQUILIBRIUM ANALYSIS	201
	7.3.1 Introductory Theory	204
	7.3.2 Equivalent Truss System	217
	7.3.3 Equivalent Element Stiffness	228

7.3.4	Assembly of Global Stiffness Matrix	232
7.3.5	Solution for Nodal Displacements	235
7.3.6	Determination of Contact Forces	243
7.4	CONCLUSIONS	248
CHAPTER 8	APPLICATIONS	
8.1	INTRODUCTION	250
8.2	MOTION OF BROKEN ICE	254
8.2.1	Rivers and Channels	255
8.2.2	Open Water	263
8.3	MOTION OF GRANULAR MATERIAL	269
8.4	CONCLUSIONS	273
CHAPTER 9	CONCLUSIONS AND RECOMMENDATIONS	
9.1	CONCLUSIONS	275
9.2	RECOMMENDATIONS	282
CHAPTER 10	REFERENCES	284
APPENDIX		
APPENDIX A		289
APPENDIX B		291
APPENDIX C		294
APPENDIX D		298
APPENDIX E		305
APPENDIX F		308

LIST OF FIGURES

Figure	Page
1.1 An example of a rigid body (a), and a flexible (b) system.	6
1.2 An example of a coupled (a) and separated (b) systems.	7
2.1 A typical example of a physical domain (a) and its model representation (b)	31
2.2 Sample clusters for the identification of rigid bodies.	35
2.3 A cluster of disks (a), and the corresponding graph (b).	39
2.4 An example of a cluster (a) and the corresponding topological tree (b).	41
2.5 An illustration of a distance event.	49
2.6 A diagram for the division of an interval using interpolation.	52
3.1 Definition of chosen generalized coordinates for a disk in contact with a line segment (a) and a disk (b).	75
3.2 Definition of generalized coordinates for a complex rigid body.	76
4.1 A schematic diagram for the rigid body generation.	80
4.2 Correlation between $F(x)$ and $f(x)$.	83
4.3 A graphical representation of the mapping of $(0,1)$ onto x -domain.	94
4.4 Discretization of the probability density function $f(x)$.	95
4.5 Sampling of the PDF by a rejection technique	100

4.6	Typical PDF's suitable (a) and unsuitable (b) for the application of rejection sampling technique.	103
4.7	Block diagram of the generation routine.	106
4.8	Sampling for the position of a rigid body.	110
4.9	Overlap between two disks.	117
4.10	Overlap of a disk with a line segment.	118
4.11	Definition of the overlap zone and overlap intervals.	121
4.12	Diagram for evaluating the surface density.	125
4.13	Different shapes of rigid bodies, triangle (a), and square (b).	128
4.14	A polygon type two-dimensional random shape.	133
4.15	A random shape rigid body created from disks of random size.	134
4.16	An example of creating a random shape rigid body.	135
4.17	Calculation of coordinates of the center of a new disk being added to the rigid body.	138
5.1	A schematic diagram of a physical domain for the Laplace equation.	157
5.2	A panel representation of a body.	160
5.3	A diagram for calculating a velocity induced by a source line segment.	162
5.4	A schematic diagram for calculation of the influence coefficients.	164
5.5	A panel representation for the cylinder.	168
5.6	Panel representation of an "L" shaped channel.	170
6.1	An illustration of the frictional forces.	181

7.1	An example of tensile (a) and compressive (b) contact forces.	192
7.2	Diagram for determination of contact forces for a disk with one point of contact.	197
7.3	Diagram for determination of contact forces for a disk with two points of contact.	199
7.4	An example of a disk conglomeration (a), and the corresponding equivalent truss system (b).	202
7.5	A schematic diagram for the formulation of the matrix equations.	205
7.6	Physical interpretation of the coefficients of matrix K.	207
7.7	A diagram for evaluating a force in a single element.	211
7.8	A definition of the elements in the equivalent truss system.	218
7.9	Definition of local systems of coordinates and corresponding displacements.	220
7.10	A diagram for evaluating the equivalent element stiffness.	228
8.1	Control area of the section of Liard river.	256
8.2	An illustration of various surface densities.	259
B.1	Diagram for calculating T.	291
D.1	Block diagram for the generation routine.	298
E.1	A schematic diagram for the formulation of the continuity equation.	302
E.2	A schematic diagram of momentum balance.	305

NOMENCLATURE

$\vec{a}$	-	Acceleration of a Particle
a	-	Coefficient
A	-	Area
A_{eff}	-	Effective Area
b	-	Parameter (increment)
$\vec{b}$	-	Body Force per Unit Mass
C	-	Concentration of Solids
C_D	-	Coefficient of Drag
CRB	-	Complex Rigid Bodies
d	-	Distance
DAE	-	Differential Algebraic Equations
E	-	Random Event
f	-	Function; Variable
$f(x)$	-	Probability Density Function
$\vec{F}$	-	Force Vector
$F(x)$	-	Cumulative Distribution Function
$\vec{g}$	-	Gravitational Acceleration
g_{ai}	-	Coefficients of Matrix
I	-	Incidence Array; Moment of Inertia
k	-	Number of events
k_s	-	Scaling Factor
l	-	Length
L	-	Generalized Inertia Forces; Length

m	-	Mass of a Particle
$\vec{M}$	-	Moment
MBS	-	Multi-Body Systems
$\hat{n}$	-	Unit Normal Vector
n_c	-	Number of Points of Contact
n_d	-	Number of Disks in Cluster
p	-	Pressure
P	-	Probability
PDF	-	Probability Density Function
PRN	-	Pseudo-Random Number
q	-	Generalized Coordinates
Q	-	Generalized Force
$\vec{r}$	-	Positional Vector
$\vec{R}$	-	Reaction Force
R	-	Radius
Re	-	Reynolds Number
RMBS	-	Rigid Multi-Body Sub-Systems
S	-	Surface
t	-	Time
T	-	Kinetic Energy of the System; Random Variable of Time
$\vec{u}$	-	Velocity Vector
V	-	Volume
x, y	-	Local Cartesian Coordinates
X, Y	-	Global Cartesian Coordinates
X_o	-	Initial Seed

W	-	Work
z	-	Overlap; Distance
Z	-	Generalized Inertia Forces
α	-	Dependent Coordinate
$\vec{\alpha}$	-	Angular Acceleration Vector
β	-	Angle
Γ	-	Space
δ	-	Increment
Δ	-	Nodal Displacement
ϵ	-	Coefficient of Restitution
ϵ_{prs}	-	Permutation Constant
η	-	Coordinate; Random Variable; Number
$\vec{\theta}$	-	Angular Displacement Vector
λ	-	Lagrange Multiplier; Poisson Constant; Second Viscosity coefficient
μ	-	Viscosity; Mean Value
ξ	-	Random Number; Coordinate
Π	-	Force Potential
ρ	-	Density
σ	-	Standard Deviation; Source Strength
$\mathcal{F}$	-	Transformation Function
ϕ	-	Potential Function
ω	-	Vorticity Vector; Angular Velocity

CHAPTER 1

INTRODUCTION AND OBJECTIVES

1.1 PROLOGUE

A system which consists of a number of independent and identifiable rigid bodies is termed a multi-body system. A system of rigid bodies in which the individual bodies can move freely relative to one another, is called here a multi-body system with variable topology. The notion of variable topology is introduced to indicate the variability of geometrical arrangements (inter-connectivity) between the bodies. This is a distinctive feature of the multi-body systems treated here. Examples of such systems include a conglomeration of rocks, a sediment accumulated by a river, or broken ice in a channel. Multi-body systems can also be characterized by fixed inter-connectivity, in which the individual components are constrained relative to one another. Such systems are termed as multi-body systems with fixed topology. A clock mechanism or a robot are examples of such systems.

One could propose an endless number of examples of multi-body systems, and an infinite array of the processes that

involve such systems. The importance of the ability to analyze their behaviour, becomes apparent.

Multi-body systems subjected to external loads, move and deform. In general, the behaviour of such systems, although, just like for any other entity, subject to certain universal laws, as for example Newton's laws of motion, conservation of energy, or Hooke's law of elasticity, is mostly governed by the system parameters and the character of the external loads applied. The external loads acting on the system are dependent on the environment to which the system is exposed, and therefore can usually be established. The governing universal laws can also be readily identified. However, the mathematical formulation, i.e. the implementation of the universal laws to the given system, and the subsequent set up and the solution of the resulting equations describing the system's behaviour, constitute the major problem, and are an art in themselves.

The complexity of the behaviour of many multi-body systems, is perpetuated by the interactions that occur among the individual bodies. The larger the number of component bodies, the more interactions occur, and hence, the more complex the behaviour. The interactions among the individual bodies which, among other things, include friction, adhesion, crushing, etc., may, in turn, be dependent on the environment

to which the system is subjected. Moreover, there also are the interactions between the bodies and the environment itself to consider. Consequently, the mathematical formulation of the process, which the given system is undergoing, becomes immensely complicated, and very difficult to implement.

The implementation of the mathematical description almost always requires a number of simplifying assumptions to be imposed. Consequently, the description is not a true representation of the process, but it can usually be considered as close enough for all practical purposes. Hence, the formulation is often referred to as a mathematical model, and the process of analyzing the behaviour of the system as modelling.

Keeping in mind the complexity inherent in many systems it is not surprising that no serious efforts to model them were made until recently, with the developments in digital computers. In general the best that can be hoped for is to develop a model represented by a set of equations in such a way that it reflects the process (which a given system is undergoing) as closely as possible, and with as few assumptions as possible.

The purpose of the present research is the development of a computer model that can be used in analyzing a variety of

the multi-body systems. In this chapter the types of systems which can be analyzed by the present method will be identified. The mathematical model and its computer implementation will be presented in the subsequent chapters.

1.2 GENERAL

Multi-body systems encompass a large variety of different systems. The systems can represent a bridge structure composed of truss, beam, and plate rigid bodies, or a conglomeration of gravel in a river bed, with pieces of debris as separate rigid bodies, or perhaps, collections of ice floes in the sea being pushed against an off-shore drilling platform. All these systems, although different in character, have one thing in common, namely that all the component bodies can be considered rigid. By rigid, it is understood that the deformations of the individual bodies are negligible with respect to their overall dimensions. The above statement constitutes a fundamental assumption that will be used throughout the present thesis. Therefore, any change in the geometry, or the topology of the system is attributed only to the movement of the individual bodies, and not to their deformations.

The multi-rigid body systems can be classified as:

Rigid systems. A system is rigid when the component bodies within the system have zero degrees of freedom. This implies, that since the motion of the bodies within the system is restricted, the overall geometry of the system

remains unchanged. The individual rigid bodies can deform under external loads, however, a position of any given body with respect to all other bodies in the system does not change. An example of such system would be a bridge structure.

Flexible systems. A system is flexible if it is not rigid.

For this class of systems relative motion among the individual rigid bodies may occur. Consequently, the topology of such a system may change as it is undergoing a given process. An example of a flexible system would be a link chain.

Figure 1.1 below shows simple examples of a rigid and a flexible multi-body systems.

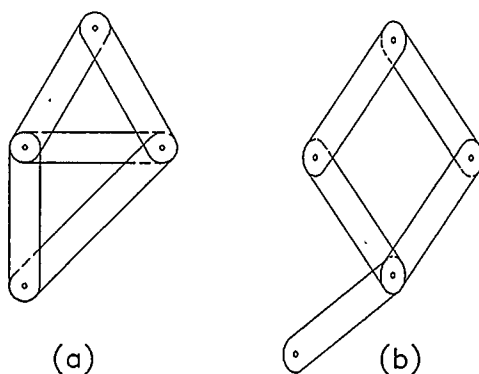


Figure 1.1. An example of rigid (a), and flexible (b) systems.

The systems can also be classified according to the constraints among the bodies making up the system. Consequently, we can identify:

Coupled System. In this system all the component rigid bodies are connected (coupled) to one another. In other words a coupled system has no bodies which are not connected to any other bodies in the system.

Separated System. A separated system is defined as one, in which none of the component bodies are connected to any other bodies in the system.

Figure 1.2 shows examples of coupled and separated multi-rigid body systems.

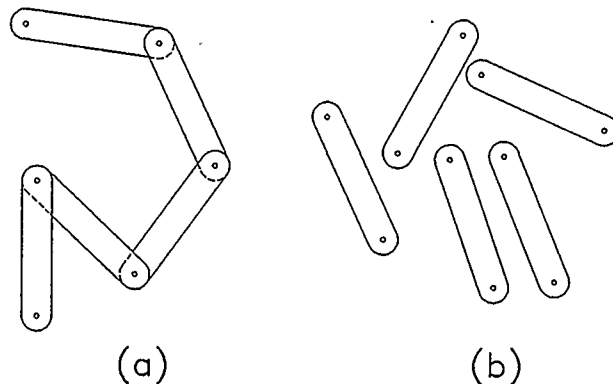


Figure 1.2. An example of coupled (a) and separated (b) systems.

From the above definitions it becomes clear that a rigid system must be coupled, whereas a flexible system may be either coupled or separated. Also an important characteristic of flexible systems is the presence of one-sided constraints, wherein a motion of a body along a certain orientation is restricted in only one direction.

Any multi-body system can be represented as a combination of different type sub-systems listed above. Let us consider an example of an automobile. The frame and the body comprise a rigid sub-system, whereas the wheels and the components of the drive train can be considered a flexible coupled sub-system. The passengers in the vehicle, if they are not wearing the seat belts, would then be represented by the separated sub-system.

Another example that can be considered is that of a granular material which moves through a chute. If the material is assumed to be cohesionless then clearly such a system will always remain separated. If on the other hand, the material possesses cohesion then some particles or debris may adhere to others, and consequently, coupled flexible and rigid sub-systems may form.

The ability to identify and classify different sub-systems in a given multi-body system is very important and can

lead to great simplifications in the mathematical model describing the system. To demonstrate this point we will consider the number of equations needed to describe the motion of a body. In two dimensions every body in a separated sub-system has three degrees of freedom, two translational and one rotational. Consequently, for every body in the sub-system there are three corresponding equations of motion. Let us now suppose that two of the bodies are connected (coupled). The number of degrees of freedom for two coupled bodies is four, instead of six for two separated bodies. Hence, the number of equations of motion is reduced by two. If the given sub-system is rigid, then regardless of the number of the bodies comprising it, the number of the degrees of freedom for it is three, and hence, only three equations are needed to describe its motion. The advantage of being able to identify different sub-systems becomes apparent, especially when dealing with large systems.

1.3 OBJECTIVES

The objective of the present study is the development of a computer model to simulate the movement of planar multi-body systems with variable topology under the action of specified external forces.

There are no restrictions placed on the character of the system itself, which suggests that the system can be composed of any types of sub-systems. In the following chapters a multi-body system with variable topology will be, for the sake of simplicity, referred to as a multi-body system (MBS).

It is also intended that the modelling can be done for systems which are submerged in a fluid medium.

There is, however, a restriction placed on the shapes of rigid bodies. Namely, that the rigid bodies can be approximated by disks. This restriction effectively suggests that the model is most suitable for analyzing systems which involve rigid bodies with comparable dimensions. The types of processes that the present model is most suitable for include: a motion of granular material (cohesionless or not), a flow of solid-fluid mixtures, ice transportation in open seas or rivers and channels, etc..

The subject of modelling of systems of this type will be introduced in detail in Chapter 2. Also in this chapter the methodological foundations of the present model will be established. The subsequent chapters will deal with presentation of the theory and its implementation on a digital computer.

CHAPTER 2
MODELLING OF THE MOVEMENT OF MULTI-BODY SYSTEMS
SUBJECT REVIEW

2.1 INTRODUCTION

As was stated in Chapter 1, the present research is directed towards the modelling of the flexible multi-body systems, which could be both coupled and separated, and in which the individual rigid bodies can be approximated by spheres in three dimensions or disks in two dimensions.

Most of the research on modelling of such multi-body systems has been concentrated in the areas of the mechanics of granular materials [2],[5],[19],[20],[22],[24],[26],[30], soil mechanics [9],[29],[33], and more recently in ice mechanics [1],[32],[35],[36], where processes such as ice transportation in rivers and channels, ice jams, or ice off-shore structure interaction, became of great interest.

In the bodies of water near the polar regions, ice floes driven by the forces of wind and current, can exert significant forces on such offshore structures as drilling

platforms and lighthouses. In the rivers and channels in freezing temperature zones, ice floes moving with the current may accumulate around the bridge piers, thus imparting loads on them. In such instances, the ability to estimate ice loads on the in-water structures, and to determine the regions of ice accumulation, becomes imperative for any design work.

In geotechnical engineering, for example, the designers may be faced with the problem of predicting the soil resistance to the movement of anchors and foundations.

A design of a chute-conveyer system for transporting gravel, may require a knowledge of the expected transporting capacity of the equipment. Or perhaps, the critical transport rate, at which the jam-up in the equipment occurs, needs to be evaluated.

The above are some examples of the processes and instances in which the analysis of the motion of flexible multi-body systems is of great importance. Due to the high cost and relatively low availability of both full and small scale experimental modelling, there seems to be an increased interest in the mathematical modelling of such systems.

Moreover, in recent years with the considerable improvements in the speed and the capacity of digital

computers, and their increased ability to handle large systems, there is a strong tendency to incorporate the mathematical models into a numerical simulation. The purpose of such simulation is to provide the required information about the behaviour of the system, from the initial set of data and process conditions.

In this chapter various existing mathematical models will be discussed. Subsequently, the methodology of the present model will be introduced. The following chapters will deal with detailed presentation of the mathematical model and its incorporation into numerical simulations.

2.2 TECHNIQUES OF MODELLING MULTI-BODY SYSTEMS

A REVIEW OF VARIOUS EXISTING MODELS

Perhaps the first attempt to analyze the motion of the solid system in a fluid was made by Einstein in 1906. Einstein studied the effect of dispersion of solid grains on the shear resistance of fluids. His analysis was limited to the process involving small spheres with concentrations small enough to neglect the effects of one grain on another. Einstein concluded that such liquid-solid mixture can be treated as a liquid with its viscosity modified to account for the presence of the solid particles. Namely, he anticipated that viscosity increases due to increased viscous dissipation in the presence of solids, that is,

$$\mu_{eff} = \mu (1 + 2.5C)$$

where μ is the viscosity of the fluid and C the concentration of solids.

The simple approach of considering the solid-liquid mixture as a continuous medium with modified parameters, which was initiated by Einstein, became the basis for the majority of the models of the granular type systems. It was not, however, until the 70's when the full use of the continuum

theory was made with the improved power of digital computers, which themselves provided the only hope of obtaining a solution to the continuum equations.

In 1954, Bangold [5] investigated a one-dimensional flow of uniformly dispersed granular material in fluid under shear. This was perhaps the first attempt in analyzing granular systems in which the effect of particle interaction was considered. Bangold, besides a constant shear strain rate, also assumed a constant kinetic energy density (uniform dispersion of solids), and no relative velocity of fluid with respect to the solid particles. He identified two basic flow regimes. The first regime for which the effects of grain inertia are negligible compared to the effects of fluid viscosity, would correspond to the flow with low shear rates and low solid concentrations. He concluded that in such instances the solid-liquid system could be treated as a liquid medium, and hence the theory postulated by Einstein could be applied. The second flow regime (identified by Bangold) corresponded to the rapidly sheared flow in which the effects of the grain inertia dominated. He proposed that in this flow regime the momentum transfer between colliding solids dominates the stress producing mechanism. He analyzed the particle collisions in a scheme of rigid wall reflection, wherein particles from one layer reflect off the particles in the adjacent layer in the same way that they would reflect off

the solid wall. He argued that because both the momentum exchanged in the collisions and the frequency of collisions are proportional to the mean shear rate, the shear stress must be proportional to the square of the mean shear rate. He then proceeded to determine the constants of proportionality experimentally.

The above model for granular materials assumes that there is no relative velocity between the solids and the surrounding fluid. Consequently, the effect of interstitial fluid is neglected. Inevitably, there appears to be some doubt regarding the proposed mechanism for collisions among the particles on which the entire model is based. Nevertheless, Bangold's work created a basis for many recent models.

In the seventies, there appeared two basic approaches to the modelling of the mechanical behaviour of granular materials. The first approach was based on the so-called microscopic or particulate theory. This approach considers an ensemble of particles of finite size (typically idealized as rigid spheres of uniform size), and attempts to deduce the laws governing the mechanical behaviour of the entire ensemble [5]. It cannot, however, be well adapted to obtaining the quantitative results as it depends greatly on the configuration of the particles.

The second method is referred to as the macroscopic approach [18] and is based on considering the granular material as continuous medium. Such an approach can more readily provide the quantitative results; however, it loses the concept of individual solid particles, and fails to incorporate the inter-particle interactions. This approach can also be applied only to systems which are at, or near to the closely packed state.

Numerous researchers combined the two approaches into a so-called mixed approach [2],[22],[26],[30], which involves obtaining quantitative continuum equations based on the microscopic properties of constituent particles.

Kanatani [22] proposed a micropolar continuum theory for the flow of closely packed granular materials. He set up quantitative equations based on conservation of mass, linear momentum, angular momentum, and energy. He then proceeded to determine the rate of energy dissipation due to the inter-particle friction by assuming constant material deformation rates on the inter-particle distances, and accordingly, by calculating the relative tangential velocity components at the particle interfaces.

By analogy to turbulent flow, Ogava et al. [26] noted the importance of the fluctuation velocities of the particles,

which inevitably result from the collisions among them. They introduced a term of fluctuation energy and attempted to determine it from the inter-particle collisions. They adapted Bangold's wall reflection scheme to a three-dimensional sphere. The radius of the sphere was equal to the mean free particle path which was determined by the particle density. The points on the sphere which correspond to the neighbouring particles, were assumed to instantaneously move with the mean flow velocity. The particle inside the sphere moved, of course, with the fluctuation velocity relative to the sphere. Ogava et al. assumed that during the collisions, a fraction of the particles adhered to the spheres with the remainder reflecting off them, with a loss of energy. They then proceeded to determine the total rate of change of fluctuation energy by averaging over all possible collisions, assuming equal reflection probability in all orientations.

Ackermann and Shen [2],[30], came up with a similar model additionally including the effect of the interstitial fluid and the mechanical properties of the solid particles on the rate of change of fluctuation energy. They suggested that the rate of dissipation of fluctuation energy depends on the frequency of inter-particle collisions which, in turn, depends on the physical properties of the solid particles and the interstitial fluid.

In the early 80's new models appeared in which the problem of flow of granular material was put in the context of the kinetic theory of gasses [19],[24].

In ice mechanics, Ackermann and Shen, [1], proposed a continuum method of modelling the process of broken ice transportation in rivers and channels. They considered the case of closely packed ice floes moving on the water surface. Ackermann and Shen treated the ice floe ensemble as a continuous medium and derived the momentum equation in the direction of the river flow, in terms of the stresses in the ice. They then used Bangold's [5] stress creation mechanism to set up the equations of motion. The numerical solutions to the equations were obtained over large range of variables and parameters which were later correlated through experimental observations. The model, however, failed to include other stress forming mechanisms, like inter-floe friction, adhesion, or crushing. Moreover, the model cannot accommodate the variability in the shapes and sizes of the ice floes as well as the presence of voids within the ice cover, the attributes which would commonly be present. Consequently, it is doubtful whether, without constant experimental verification, the model could be used to obtain quantitative results for a general case.

Several models based on the continuum theory have been

discussed. The number of different models is probably as large as the number of researchers involved with the subject. The models vary in complexity, and in the number and the character of simplifying assumptions. All the models, however, originate from the same point. They carry the assumption that the granular material can be treated as continuous medium. Thereafter, the continuum equations of the conservation of mass, momentum (linear and angular), and energy are set up. The physical properties of the solid particles and interstitial fluid are then incorporated into the continuum equations through the assumed microscopic mechanisms of particle interactions. The continuum models can therefore be applied to the systems in which the solid bodies, represented by spheres and disks, are closely packed. These models also relate only to the processes involving high shear rate flows, as the inter-particle interactions are assumed to be the dominant mechanism in stress creation.

The continuum models are not suitable for analyzing the loosely packed systems with large voids (containing no solid particles) present, or systems with non-homogenous dispersion of solid bodies, as can often be the case with many processes. Moreover, such models cannot account for the variations in particle sizes and in their physical properties. Also the inter-particle adhesion and the resulting particle clustering cannot be accounted for.

The continuum models are, also, written using Eulerian description. This means that the focus is put on certain fixed points in space through which different solid particles pass at different times, rather than on certain particles or ensembles of particles. Consequently, these models cannot be used to analyze processes in which the identification of the critical areas of solid bodies accumulation, or the predictions of the regions of jam-up, are vital.

Considering the difficulties with the application of continuum models to various processes involving the multi-body systems, it is not surprising that during the past decade, a new approach to the modelling of such systems, based on discrete analysis, appeared [9],[29],[31],[33],[35]. With the increasing power of digital computers, it became feasible to model the granular type multi-body systems as a particulate rather than continuous material. In such an approach, each solid particle (rigid body) in the system is looked at as an individual entity whose motion is followed throughout the process (Lagrangian description). The interactions among the particles are derived from their particular positions and velocities, and their physical properties, rather than by averaging procedures, as is the case in continuum models. Consequently, the motion of each particle is established at any time during the process, and hence, the behaviour of the entire system determined.

The first attempts at discrete modelling of granular assemblies were done by Cundall and Strack [9]. The model was directed at problems in geotechnical engineering, and in particular, soil mechanics. Various simulation programs were developed based on this model [29],[33]. The interest in discrete modelling of soils was bolstered by the fact that it provided the researchers with a possible tool for handling the problems which present a major difficulty for conventional continuum models, such as a considerable soil non-homogeneity, non-linear soil response, or non-linear soil-structure interaction.

The Discrete Element Method (DEM) utilized in [9],[33], is a two-dimensional model in which solid bodies are represented by disks. The model is based on setting up the equations of motion from Newton's Second Law, for each individual disk. The forces and moments acting on each disk include both the external and the interaction loads. The interaction between the individual disks is modelled using a spring and damper system. Therein, the shear and normal contact forces are determined from the amount of overlap that appears among the disks during the simulation, and the rate of change of the overlap. Once all the loads (external and contact) acting on each individual disk, are evaluated, the linear and angular acceleration terms are computed from the equations of motion, and subsequently integrated over a small

time interval to yield new positions and velocities of the particles.

This model was applied, with some success, to certain segregation and anchor pull-out problems. It can handle Coulomb friction and non-homogeneity of the material as different spring-damper systems can be applied at different contact points. It can also accommodate the non-linear behaviour of the material through nonlinear springs and dampers. The model, however, seems very sensitive with regard to the time step, and the spring and damper constants, since during the simulation the overlap among the particles is a determining factor in the computation of forces. For larger time steps and stiffer spring-damper systems, unrealistically high contact forces may result, while smaller time steps and softer springs may not allow for the proper accommodation of the impact forces among the solid bodies.

A similar approach was applied in ice mechanics in [17]. In this model the ice floes and ice sheets were represented by finite elements. The formulation was based on the solution of the dynamic equilibrium equations with a set of decoupled orthogonal nodal equations for each element. In the model the element interaction occurred through the boundary forces.

A different approach to discrete analysis was postulated in [35],[38] and further developed in [31]. The mathematical formulation in this approach is based on the Lagrangian dynamics. The rigid bodies are approximated by disks. Here, rather than setting up the dynamic equilibrium equations for each disk individually, the equations of motion are derived for the entire disk conglomeration in terms of generalized coordinates. The interactions among the rigid bodies are identified through the so-called constraint equations. The system of differential equations of motion together with the algebraic constraint equations, is solved for the generalized coordinates and the unknown constrain forces. Consequently, the new positions of all the bodies in the system are determined.

The advantage of such a model is in the fact that the momentum transfer among the rigid bodies during collisions can be readily included. Also, the friction among the bodies can be handled without much difficulty, since the frictional forces depend on the constraint forces, which, as stated above, are computed.

The method has a disadvantage however, which is the system of differential algebraic equations (DAE). Obtaining a solution to a large system of DAE's presents a significant numerical difficulty [11],[12],[13]. In fact, not much is

known about the general methods of solution, and problems with numerical instability can be frequently encountered.

The numerical model for analyzing the behaviour of granular type multi-body systems developed in the present research work is based on the Lagrangian approach. As will be seen in the following chapters, certain ideas and simplifications are incorporated into the model to reduce the number of DAE's, and to improve the stability of the solution.

2.3 PROPOSED TECHNIQUE OF MODELLING

The present research is directed at the development of a two-dimensional computer model for analyzing the behaviour of flexible multi-body systems, in which the individual rigid bodies can be approximated by disks. Although, in principle, any type of flexible system can be modelled by this technique, particular attention is paid to the modelling of separated systems (Figure 1.2b). Analyzing the behaviour of such systems is difficult, and no general methodologies exist.

Since the present analysis is directed specifically towards the flexible multi-body systems, in this thesis we will refer to such systems as simply multi-body systems (MBS).

A MBS is a specific type of system. The individual rigid bodies cannot overlap onto, or penetrate one another, and hence the motion of the bodies is constrained in the direction towards one another. On the other hand, the rigid bodies can separate from other bodies, if some appropriate conditions are met, and thus their motion away from one another is unconstrained. This type of system is characterized as having the so-called one sided constraints.

It is due to the presence of one sided constraints that

the topology of MBS constantly changes with time, as the connections disappear and new connections are formed. Moreover, the moments in time when the topology of the system changes are unknown and cannot be predicted, since it is unknown for how long certain bodies will remain in contact, or how soon a given body will attach to any other body in the system. Consequently, the governing equations of motion for the system cannot be written in advance. It, therefore, becomes clear that the only feasible approach to the analysis of MBS is through computer simulations.

In this section a general methodology of the proposed numerical simulation of multi-body systems with one sided constraints will be discussed.

2.3.1 General Approach

The computer simulation takes place in a designated space, or a control volume, and a designated time frame. The control volume is conveniently chosen by the program user, and represents the area in which the simulation is to be carried out.

All the solid boundaries and obstacles inside the control volume, with which the rigid bodies may interact, are represented by their contours. The contours are discretized into a set of straight line segments, in which form they are entered into the simulation program. The solid boundary is assumed to be impregnable to the rigid bodies. In order to avoid a singularity at the solid boundary, small arcs are created at the points where the two adjacent line segments (describing the boundary) meet.

Inside the control volume there are also boundaries at which the rigid bodies enter the system, and at which they leave it. These are referred to as the generation and the exit boundaries, and are discussed in more detail in Chapter 4.

The rigid bodies are represented in the model by disks or

rigid sets of disks. The sizes of the rigid bodies are either predetermined or randomly picked by the program according to a specified distribution function, which is supplied as an input. Similarly, the rigid bodies can be placed inside the control area in specified positions, or generated at the generation boundary at random time intervals and random locations. The methodology of the generation of rigid bodies is discussed in detail in Chapter 4.

A typical representation of the physical domain in which the simulation of the motion of MBS is to be carried out, as used by the model is shown in Figure 2.1.

All the disks present in the control volume are acted upon by the external forces. The character of the forces present in the system must be identified prior to the start of the simulation procedure. The external forces can be, for example, drag forces if the given MBS is moving in a fluid medium, or gravity forces if the MBS represents a conglomerate of dry granular material sliding down a chute.

In a typical simulation procedure the disks (rigid bodies), which are either placed or generated, move through the control volume due to the action of the external forces. The disks interact with other disks and with the obstruction line segments. The motion of the disks is modelled with the

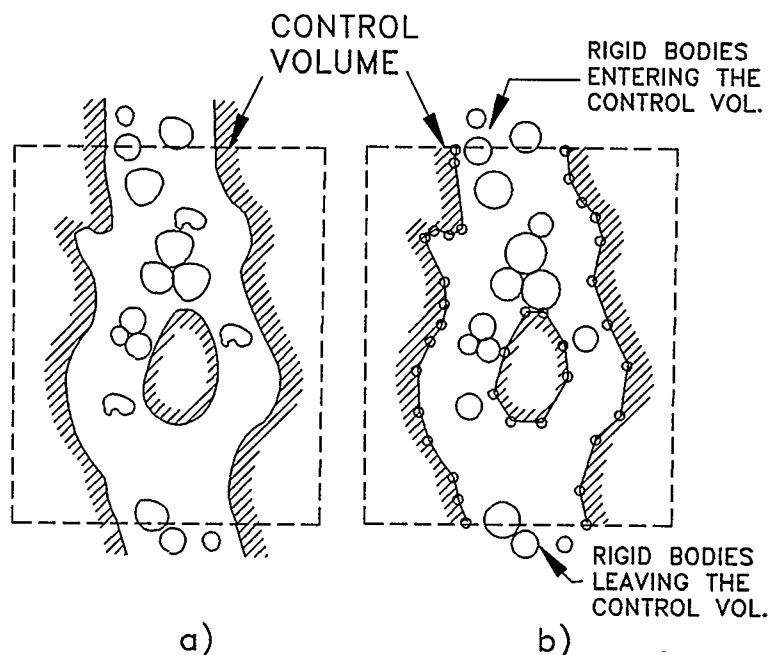


Figure 2.1. A typical example of a physical domain (a) and its model representation (b).

equations of motion which are derived from the principles of Lagrangian dynamics of rigid bodies. The mathematical formulation of the equations of motion was developed in [31] and is briefly presented in Chapter 3. At each time step during the simulation, the equations of motion for the entire disk ensemble are established. These equations are then integrated over a specified time interval, and the new positions for the disks, at the new time step, are obtained. At this time, again, the external forces are determined and a new set of equations of motion is set up, and the procedure repeated.

At each time step the equations of motion are set up according to the topology of the MBS. During the simulation process the topology of the system will undergo constant changes. Some disks will collide with one another, while others may adhere to the solid boundary or other disks. Consequently, the topology at each time step must be determined. The topological analyses of MBSs were proposed in [35] and [38], and developed in [31]. This will be briefly discussed in the next section.

2.3.2 Topological Analysis

The system under consideration consists of so-called base elements. These are disks representing the rigid bodies, and the line segment, denoting the solid boundary. Each disk is assigned a radius and a mass. A position of a disk is described by the coordinates of its center. Such a base element is mobile which means that its position may change in time. A line segment is a base element which is considered fixed in space. The position of the element is given by the coordinates of its endpoints.

In the present simulation program, it is assumed that the base elements cannot overlap. This implies that any two given line segments cannot cross, a disk cannot penetrate into a line segment (solid boundary), and two disks cannot overlap at any time during the simulation. The base elements can, however, become attached, which means that they have a point of contact. From the definition of the base elements, it becomes clear that any two elements can have at most one point of contact.

A group of disks, such that each disk in that group has at least one point of contact, is termed a cluster. Hence, any MBS is a combination of clusters and single bodies.

A disk which is in contact with a line segment is termed a root disk. A root disk can belong to a cluster, in which case the cluster is said to be rooted.

A single disk is referred to as a simple rigid body. A complex rigid body is a cluster of disks arranged in such a way that it will not change its shape provided that the connectivities among the disks are maintained. Using the analogy to a two-dimensional truss system, we can write the following conditions.

- 1) A cluster of disks is not rigid if and only if

$$2n_d - n_c > 3$$

where n_d is a number of disks in the cluster, and n_c is a number of points of contact.

- 2) A cluster of disks is rigid if and only if

$$2n_d - n_c \leq 3$$

- and
- a) there are no disks with one point of contact,
 - b) any disk having two points of contact cannot be touching any other disk with two points of contact.

Figure 2.2, below, exemplifies the above conditions for the rigidity of the clusters of disks.

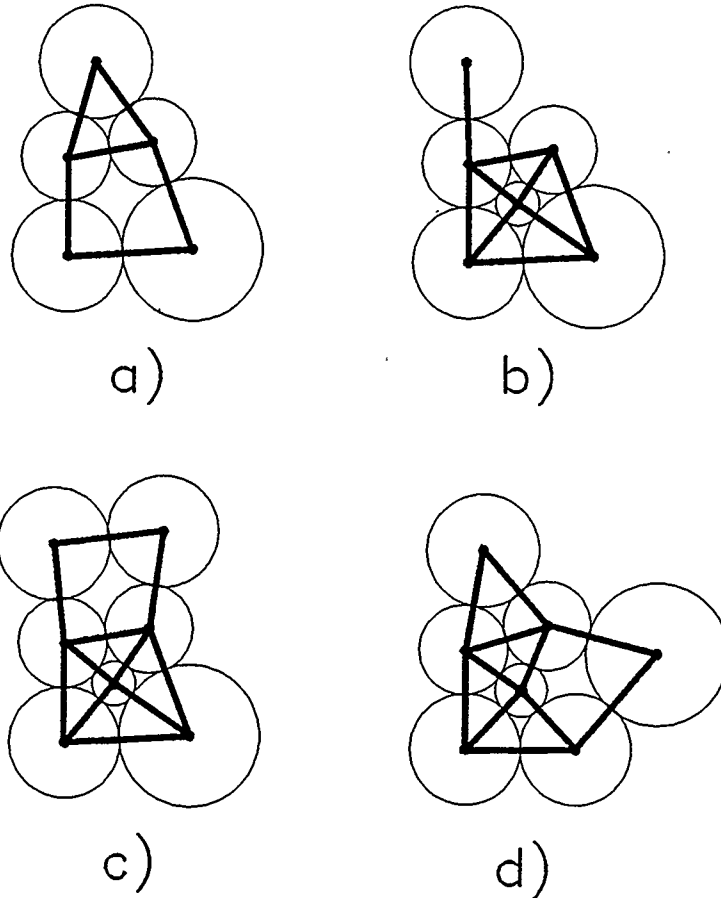


Figure 2.2. Sample clusters for the identification of rigid bodies.

In Figure 2.2 for cluster (a) $n_d = 5$, $n_c = 6$, and thus condition 1) is satisfied. Hence, the cluster is not rigid. This can also be verified by inspection.

For cluster (b) $n_d = 6$, $n_c = 9$, and evidently condition

2) is satisfied which indicates that the given cluster can be rigid. However, there is one disk which has only one point of contact. Consequently, condition 2a) is not satisfied, and thus, as can be verified by inspection, the cluster is not rigid.

For cluster (c) in the figure $n_d = 7$, $n_c = 11$. Condition 2) is satisfied. Part b) of the condition is not satisfied, as there exist two disks in the cluster, having two points of contact, which are also in contact with another disk having only two points of contact. Hence, cluster c) is not rigid.

For cluster (d) $n_d = 7$, $n_c = 11$. This cluster is rigid. Condition 2) is satisfied together with additional conditions a) and b). There are no disks with one point of contact, and the disks with two points of contact are not touching any other disks with two points of contact. The rigidity of this cluster can be easily verified by inspection.

The present simulation program is capable of generating both the simple and the complex rigid bodies. Chapter 4 discusses the generation of the rigid bodies in detail.

It is assumed that the simple rigid bodies (disks) have only two degrees of freedom. In other words, the rotation of a disk with respect to its center is neglected, as it does not affect the geometry of the entire system.

The complex bodies (rigid clusters) possess three degrees of freedom, two translational and one rotational. Consequently, three coordinates are needed to describe the position of a complex body, for example, two cartesian coordinates of the center of gravity, and the angle of rotation between an axis fixed with the cluster and the X-axis. This, of course, holds true regardless of the number of disks composing the cluster.

The observation that a rigid cluster has only three degrees of freedom is an important one. The number of equations needed to describe the motion of a MBS is equal to the number of degrees of freedom. Consequently, the identification of all the clusters which during a given time step can be considered rigid, can greatly reduce the number of equations of motion required. This brings us to the concept of rigid multi-body subsystems (RMBS). This concept was postulated in [37] and developed in [31].

RMBS are the bodies of the same type as the complex rigid bodies (CRB) in the sense that provided that all the connections are maintained the entire configuration will remain unchanged. Unlike the CRB's in which the connectivities among the disks are maintained throughout the simulation process, the connections among the disks composing a given RMBS may be deleted during the simulation, if a

certain condition is met. Namely, that the contact force is tensile. Also, during the simulation new disks may join the RMBS, once the appropriate connections are created. A condition for a new disk to be added to a RMBS is that it forms a connection with any two disks already present in the RMBS.

To summarize the above definitions, we can say that rigid multi-body sub-systems are rigid clusters whose respective topologies, although unchanged over some time interval, may change during the simulation process, while complex rigid bodies are rigid clusters whose topology remains constant throughout the simulation. Incidentally, an RMBS can be composed of several CRBs and a number of single disks.

Having identified all the different bodies and elements, we can proceed with the description of the topological analysis on which the mathematical formulation of our model is based.

As mentioned above any MBS is composed of individual disks and clusters of disks. A cluster as an object is characterized by the number of disks, the coordinates of their centers, and the coordinates of the points of contact. This data describes the topology of the system. If the pairs of contacting disks are represented by the line segments joining

their centers, then a cluster can be uniquely represented by a graph. Figure 2.3 shows a cluster with a corresponding graph defining its topology.

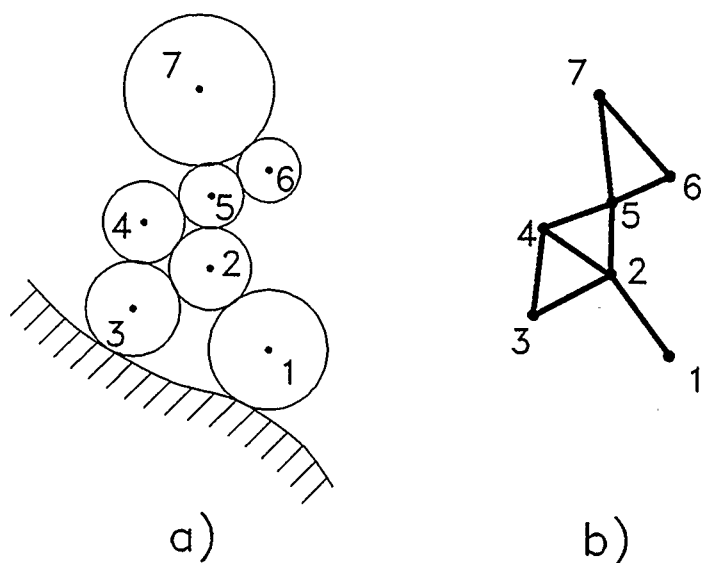


Figure 2.3. A cluster of disks (a), and the corresponding graph (b).

The centers of the disks composing the cluster are referred to as the nodes, while the line segments joining the respective centers as the edges. A base node is identified for each cluster, with respect to which all other nodes (centers of disks) can be defined. In the case of a cluster adhering to the boundary line segment, a root node (root disk) becomes a base node. A graph of a cluster of disks is conveniently stored in an incidence array $[I]$. The entries of

this array are either one or zero. An entry of 1 indicates that a connection exists between the two respective disks. Zero is equivalent to no connection. Hence, if $I_{ij} = 1$, then connection exists between disk i and disk j . For the example in Figure 2.3 the incidence matrix would be as follows:

$$[I] = \begin{bmatrix} \cdot & 1 & 0 & 0 & 0 & 0 & 0 \\ 1 & \cdot & 1 & 1 & 1 & 0 & 0 \\ 0 & 1 & \cdot & 1 & 0 & 0 & 0 \\ 0 & 1 & 1 & \cdot & 1 & 0 & 0 \\ 0 & 1 & 0 & 1 & \cdot & 1 & 1 \\ 0 & 0 & 0 & 0 & 1 & \cdot & 1 \\ 0 & 0 & 0 & 0 & 1 & 1 & \cdot \end{bmatrix}$$

The diagonal elements in the above matrix are not indicated as they are used to store a number of different parameters.

A path is defined as the shortest route along the edges of the graph from the base node to the given node. A set of paths constitutes what is referred to as the topological tree. The edges spanning the respective nodes in the topological tree are described as branches.

For the cluster of Figure 2.3, the corresponding topological tree is shown in Figure 2.4.

A topological tree can be represented numerically in terms of a two-dimensional array $[T]$. This array, unlike the

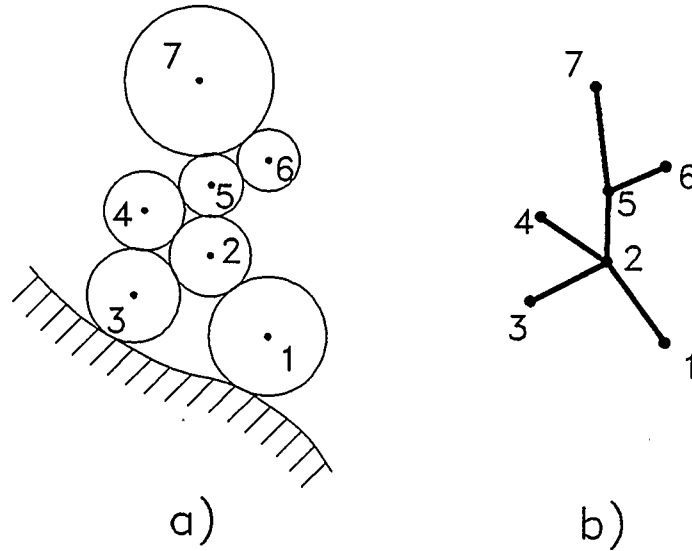


Figure 2.4. An example of a cluster (a) and the corresponding topological tree (b).

incidence array $[I]$, is not symmetric. Its entries can be either zero or one. Each row of array $[T]$ represents a path to the corresponding disk. The entries in any given row, equal to one indicate that the corresponding node (disk) is on the path from the base node to the given node. For example, if $T_{ij} = 1$, then this means that node j is along the path from the base node to node i . For the situation shown in Figure 2.4 the topological tree array would be as follows:

$$[T] = \begin{bmatrix} 1 & 0 & 0 & 0 & 0 & 0 & 0 \\ 1 & 1 & 0 & 0 & 0 & 0 & 0 \\ 1 & 1 & 1 & 0 & 0 & 0 & 0 \\ 1 & 1 & 0 & 1 & 0 & 0 & 0 \\ 1 & 1 & 0 & 0 & 1 & 0 & 0 \\ 1 & 1 & 0 & 0 & 1 & 1 & 0 \\ 1 & 1 & 0 & 0 & 1 & 0 & 1 \end{bmatrix}$$

The two arrays $[I]$, $[T]$ are sufficient to define the connectivities in, and the topological tree of, any cluster of disks. From these two arrays the positional vector of any disk with respect to the base disk (base node) can be defined, and hence, the equations of motion can be set up.

The length of each branch on the topological tree is equal to the length of the vector spanning the respective nodes (centers of the disks). Consequently, if the positional vector of the base node of the cluster is known along with the topological tree, then the positional vector of any other node (disk) can be determined by "vectorially following" an appropriate path. For example, from Figure 2.4 a positional vector for disk (7) would be given by

$$\vec{r}_7 = \vec{r}_1 + \vec{r}_{1,2} + \vec{r}_{2,5} + \vec{r}_{5,7}$$

The above equation can also be written in terms of the two arrays, in the form that it can be implemented numerically. Namely, in a general case,

$$\vec{r}_i = \vec{r}_0 + \sum_{j=1}^{N_D-1} \left[\sum_{k=j+1}^{N_D} (I_{jk} \cdot T_{ij} \cdot T_{ik} \cdot \vec{r}_{j,k}) \right]$$

where $\vec{r}_0$ represents a positional vector of the root node of the given cluster and N_D is the number of disks comprising the cluster.

A positional vector representing a branch of the tree (such as $\vec{r}_{2,5}$) is defined by its length, which is known and equal to the sum of respective radii (for disk 2 and 5), and by its orientation, which is given by the angle between it and the x-axis. These angles are referred to as the generalized coordinates for the cluster. Evidently, a positional vector of the center of any disk in the given cluster can be defined in terms of the positional vector for the base disk and the generalized coordinates corresponding to the branches of the topological tree. The equations of motion for a cluster, and the entire MBS are, as we shall see in Chapter 3, derived in terms of the generalized coordinates.

In the above examples the topological analysis was performed on a cluster composed of disks. In a similar way, the topology of a cluster in which complex rigid bodies (CLB), or rigid multi-body sub-systems (RMBS) are present, can be analyzed. Here, every rigid body would be treated in the same way as an individual disk.

The detailed topological analysis of MBS's used by this model, utilizing the concept of rigid multi-body sub-systems, was numerically implemented in [31].

2.3.3 Events

During the simulation, every occurrence at which the topology of the system changes, is referred to as an event. A new disk entering a control volume represents an event, as does a collision between two disks or between a disk and the obstruction. A disk separating from, or joining a cluster of disks is also considered an event.

Since an event corresponds to a change in topology, it follows that the generated equations of motion are valid only between the events (i.e. the time intervals over which the topology of the system remains constant). Consequently, all the events must be detected and the times of their occurrence determined. The topology of the system must then be reanalysed and the new set of equations of motion generated.

There are three primary types of events: a disk event, a distance event, and a force event. The latter two, in terms of numerical simulation, correspond to a change in constraint conditions of the system, while the first type of event (disk event) represents an addition of a new body to the system.

A disk event is relatively easy to handle. The simulation model in the process of generating the rigid

bodies, generates the time increments between the consecutive disk events (Chapter 4). Therefore, the times of disk events are always known. Consequently, the simulation time step is adjusted in such a way that the disk event coincides with the incremented simulation time, at which point the topology of the system can be reanalysed to include a new disk.

Handling of the distance and force events is not as straight forward as that for a disk event. These events correspond to the interactions among the different elements in the system, such as collisions between disks, separation of a disk from, or adherence of a disk to a cluster, etc.. The times when such events take place during the simulation can not be predicted beforehand. They must be determined during the course of the simulation.

Typically, a force or distance event would take place at some point between two consecutive time steps at which a detection of an event can be carried out. If an event is detected over the given time interval, then the simulation is stepped back over this time interval until the instance, at which the event occurred, is determined. The simulation clock is then set back to that point in time, the topology of the new system analyzed, and the new set of equations of motion generated.

As was stated in the previous section, during the simulation the topology of the system, on which the equations of motion are based, is considered constant during short time intervals over which the equations of motion are integrated. In other words, it is assumed that during the time increments the constraints (connections) among the disks and the obstruction line segments are unchanged. A constraint is maintained until the corresponding contact force between the two elements becomes tensile and exceeds a certain predefined limit. If such a situation arises, then it is said that a force event occurred, and the corresponding constraint is eliminated. This, of course, is equivalent to breaking the appropriate connection. If one-sided constraints are modeled then the limit for the contact force is zero, which indicates that the elements are unrestrained in the direction away from one another. This would be the case if a system involving cohesionless solid bodies, was modelled. If, on the other hand, complex rigid bodies are involved, or if cohesion needs to be accounted for, then this limit could represent the tensile strength of the given connection. To determine whether a force event occurred, all the contact forces among the elements present in the system (disks and obstruction line segments) are calculated at the beginning of each time step. The computation of contact forces is discussed in Chapter 7.

The final type of event is a distance event. This event

occurs when a disk collides with another disk or an obstruction line segment. In terms of the numerical simulation, this takes place if during the simulation an overlap develops between any two elements in the system. An overlap between two disks occurs if the distance between their centers is smaller than the sum of the respective radii. An overlap between a disk and an obstruction line segment occurs if the distance from the center of the disk to the line segment is shorter than the radius of the disk.

A presence of an overlap between a disk and another element, at a given time during the simulation suggests that at some point during the preceding time interval, the disk involved must have come in contact with that element. Consequently, the motion of the disk, which was assumed unconstrained in the particular direction at the beginning of the time step, became constrained at that point in time. Henceforth, the equations of motion, which were originally assumed valid over the entire time interval, were only valid up until the time of collision. After that time, a different set of equations described the motion of the system, as its topology changed. Moreover, if a disk collides with a cluster, then the impact produced by the collision results in impulsive loading at all connections in the cluster involved. As a consequence, other connections can be broken or created. It is therefore evident that the time, at which the collision,

represented by the distance event, takes place, must be determined.

To detect a distance event, the coordinates of the center of every disk present in the system, in the inertial frame of reference, are determined at all times during simulation. From these locations a distance from the center of any given disk to another element in the system can be computed. The event is detected if that distance is shorter than the radius of the disk. Figure 2.5 illustrates a distance event between two disks.

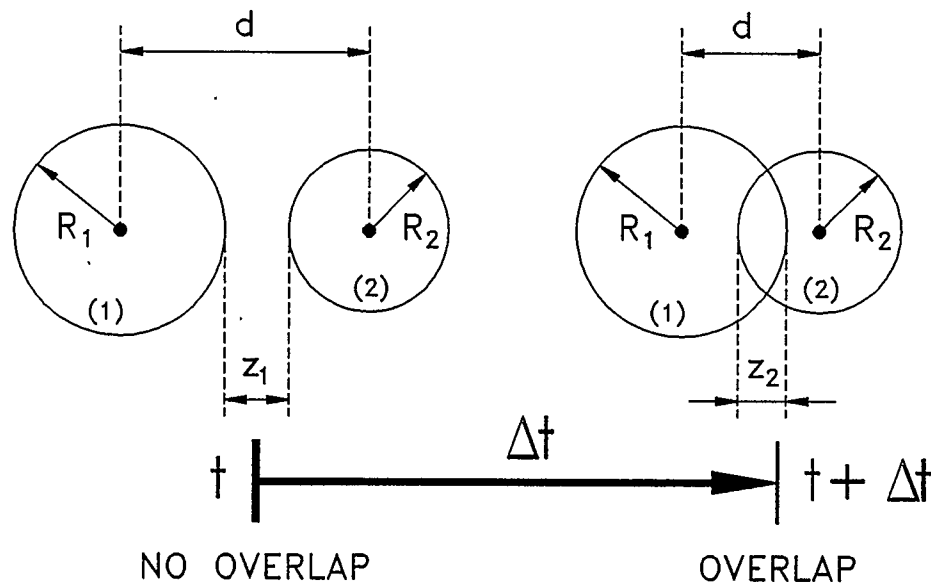


Figure 2.5. An illustration of a distance event.

An overlap is defined in terms of variable z ,

$$z = R_1 + R_2 - d$$

A negative value of the variable z indicates no overlap, whereas a positive one corresponds to an overlap.

Once a distance event is detected over some time step Δt , the exact point in time at which the event occurred, must be determined. This is done by dividing the time interval into two sub-intervals Δt_1 , Δt_2 , such that:

$$\Delta t_1 = (t, t + \Delta t_1) \quad , \quad \Delta t_2 = (t + \Delta t_1, t + \Delta t)$$

The distance event must occur during either of the two subintervals.

First, sub-interval Δt_1 is considered. The simulation clock is set back to the beginning of this time step (t), and subsequently, the simulation is carried out over Δt_1 . If an overlap is detected at time $t + \Delta t_1$, then the given distance event occurred during this time step. If such is not the case, then clearly the distance event must occur during sub-interval Δt_2 .

Having identified the time sub-interval over which the

distance event occurs, we repeat the above procedure thus narrowing the event's time frame. The whole process is carried out until the size of the final sub-interval is equal to the assumed tolerance for the time of the event. The mid point of the final sub-interval is the determined time of the distance event.

There are two methods for the division of a given time interval into two sub-intervals, that were considered for the present simulation model. The first one is based on the bisection method. Here, the interval is always divided into two equal sub-intervals (divided in half). Hence,

$$\Delta t_1 = \Delta t_2 = \frac{\Delta t}{2}$$

The second method is based on the linear interpolation. Here, use is made of the amount of separation at time t and the amount of overlap at the time at which the event was detected ($t + \Delta t$). It is assumed that the formation of the overlap is linear with time, and hence the sub-intervals should be in the same proportion relative to each other, as are the amounts of the separation and the overlap. From this condition the sizes of the sub-intervals can be determined. Using the example of Figure 2.5 the interpolation method is illustrated in Figure 2.6.

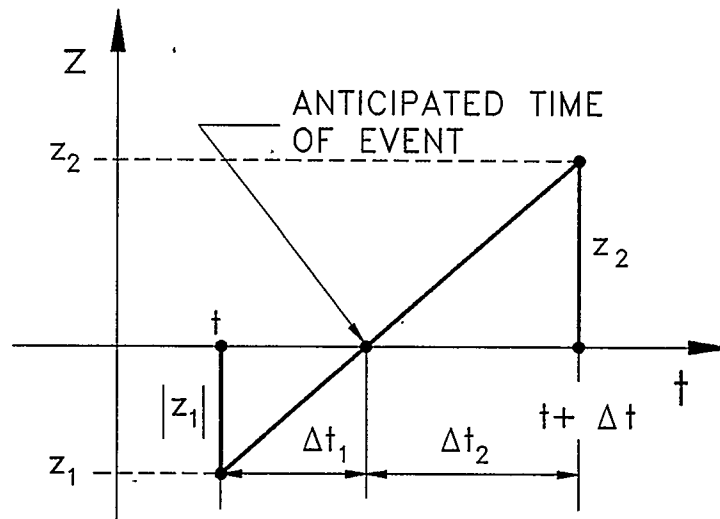


Figure 2.6. A diagram for the division of an interval using interpolation.

From the diagram we have

$$\frac{\Delta t_1}{|z_1|} = \frac{\Delta t_2}{z_2} = \frac{\Delta t}{|z_1| + z_2}$$

Consequently, the sizes of subintervals are determined,

$$\Delta t_1 = \Delta t \cdot \frac{|z_1|}{|z_1| + z_2}$$

and

$$\Delta t_2 = \Delta t \cdot \frac{z_2}{|z_1| + z_2}$$

At the present time the bisection method for dividing the time interval is used. It is however believed that the linear interpolation method would in the majority of instances yield

results faster.

The methodology of event determination and handling discussed in this section, was numerically implemented in [31] and the detailed analysis of this can be found in the reference.

2.4 CONCLUSIONS

In this chapter, the subject of modelling of multi-body systems with one-sided constraints was introduced. Two basic approaches to the modelling of such systems were identified: a continuum approach, and a discrete approach. The continuum approach can, at best, be used in modelling the systems in which solid bodies are closely packed, or can be assumed fully fluidized, since the motion of a system is described by one set of equations which are assumed applicable at any point in time. Consequently, the interactions among the rigid bodies can only be considered in a statistical sense, through the averaging procedures. It is evident that such a methodology can provide a good physical representation for systems which are characterized by a loose and non-uniform distribution of solid bodies. The difficulty in analyzing such systems lies in the fact that their behaviour is greatly influenced by relative positions of the solid bodies and the interactions among them. These, however, undergo constant changes, and thus cannot be predicted beforehand. Bearing this in mind, it becomes clear that a discrete approach combined with a time-based numerical simulation is the only viable alternative to modelling of multi-body systems. The present thesis deals with the development of simulation methodology based on the discrete approach.

During the simulation, the motion of each solid body is traced throughout the entire process, and the body's interactions with other objects in the system established. The equations of motion governing the behaviour of the system are not permanent throughout the simulation, but are constantly regenerated to accommodate the changing topology and the constraint conditions of the system. The topology of the system changes due to the changing interactions among the system's elements. The changes in interactions and their character were classified as events. Three fundamental types of events were identified: a force event which results in the breaking of a constraint, a distance event which creates an additional constraint, and a disk event which adds a new rigid body to the system. During the simulation the events are determined and the topology of the system modified accordingly.

It is believed that the present model can be used to analyze a large variety of multi-body systems with one-sided constraints. Although at the present stage the model is only two-dimensional and a number of simplifying assumptions is made, the mathematical formulation of the problem is exact. Consequently, the validity of the analysis and the results that it yields, is only limited by our ability to physically identify and interpret all the interactions present in the actual system.

CHAPTER 3

MATHEMATICAL MODEL

3.1 INTRODUCTION

The mathematical model incorporated into the present numerical simulation of MBSs is based on Lagrangian principles of solid body dynamics. The Lagrangian dynamics provides us with a set of motion and constraint equations which describe the behaviour of the system. A given set of equations is only valid over the time interval during which the topology of the system remains constant. Consequently, the constraint and motion equations are constantly adjusted and reset to accommodate the changes in the topology. The topological analysis which was discussed in Section 2.3.2 provides us with the means of conveniently generating these equations.

In this chapter a mathematical formulation of the equations of motion together with the constraint equations will be discussed. The equations of motion are given by the second order non-linear differential equations, which are commonly referred to as the Lagrangian equations of the second type. The constraint equations represent a set of algebraic equations. Together, the motion and constraint equations

comprise what is known as the set of differential algebraic equations (DAE).

It is assumed that the multi-body systems under consideration can only have geometrical constraints imposed upon them. Such systems are referred to as holonomic. In the following analysis we will limit ourselves to the consideration of only such systems.

3.2 LAGRANGIAN DYNAMICS

In this section a derivation of the Lagrangian equations of motion is briefly presented. This is done for the convenience of the reader and the completeness of the thesis. A detailed derivation of the equations can be found in any textbook on Lagrangian dynamics (see, for example [39]).

D'Alembert's principle states that any position of a system during its motion can be analyzed as a position of equilibrium by adding the inertia forces to the active forces acting on the system at that instant. The principle thus allows the application of static methods to dynamic problems.

Let us consider a system containing N particles. Using D'Alembert's principle, the equilibrium condition for each particle can be written as:

$$\vec{F}_j - m_j \vec{a}_j = -\vec{R}_j, \quad (j = 1, \dots, N)$$

where $\vec{R}_j$ are the reaction forces resulting from the constraints imposed on the system. Since the system is in equilibrium then the application of the principle of virtual work yields:

$$\sum_{j=1}^N (\vec{F}_j - m_j \vec{a}_j) \cdot \delta \vec{r}_j = 0 \quad (3.1)$$

where $\delta \vec{r}_j$ represents a set of virtual displacements. In the above equation the term $\sum \vec{F}_j \cdot \delta \vec{r}_j$ represents the virtual work done by the active forces of the system, while term $\sum m_j \vec{a}_j \cdot \delta \vec{r}_j$, represents the virtual work of the inertia forces. These will be denoted as δW , δZ respectively. We could therefore write Equation (3.1) as

$$\delta W = \delta Z$$

Let us also assume that the given system possesses n degrees of freedom (the number of particle coordinates minus the number of geometrical constraints). We can thus select a set of n independent variables q_i , referred to as generalized coordinates, such that the position of any particle in the system can be uniquely represented as a function of these coordinates and time. We write,

$$\vec{r}_j = \vec{r}_j(t, q_1, \dots, q_n) \quad , \quad (j = 1, \dots, N)$$

Since the virtual displacements $\delta \vec{r}_j$ represent the virtual differentials (fixed in time) of the positional vectors $\vec{r}_j$ we have

$$\delta \vec{r}_j = \sum_{i=1}^n \frac{\partial \vec{r}_j}{\partial q_i} \delta q_i \quad (3.2)$$

Substituting Equation (3.2) into Equation (3.1) we can write the former as

$$\sum_{j=1}^N \left[\vec{F}_j \cdot \sum_{i=1}^n \frac{\partial \vec{r}_j}{\partial q_i} \delta q_i \right] = \sum_{j=1}^N \left[m_j \vec{a}_j \cdot \sum_{i=1}^n \frac{\partial \vec{r}_j}{\partial q_i} \delta q_i \right]$$

Since the order of summation in the above expression is not important, we can perform the summation over j first. We may therefore write

$$\sum_{i=1}^n \left[\sum_{j=1}^N \vec{F}_j \cdot \frac{\partial \vec{r}_j}{\partial q_i} \right] \delta q_i = \sum_{i=1}^n \left[\sum_{j=1}^N m_j \vec{a}_j \cdot \frac{\partial \vec{r}_j}{\partial q_i} \right] \delta q_i \quad (3.3)$$

In the above equation the term on the LHS represents the virtual work done by the active forces, while the term on the RHS corresponds to the virtual work of inertia forces. We will write Equation (3.3) as

$$\delta W = \sum_{i=1}^n Q_i \delta q_i = \sum_{i=1}^n L_i \delta q_i = \delta Z \quad (3.4)$$

where:

$$Q_i = \sum_{j=1}^N \vec{F}_j \cdot \frac{\partial \vec{r}_j}{\partial q_i} , \quad L_i = \sum_{j=1}^N m_j \vec{a}_j \cdot \frac{\partial \vec{r}_j}{\partial q_i}$$

Coefficients Q_i are referred to as generalized forces. Coefficients L_i which represent generalized inertia forces can be expressed in terms of the kinetic energy of the system T ,

(see Appendix A), as

$$L_i = \frac{d}{dt} \frac{\partial T}{\partial \dot{q}_i} - \frac{\partial T}{\partial q_i}$$

Equation (3.4) is satisfied for any arbitrary set of generalized virtual displacements δq_i . It therefore follows, that the corresponding terms must be equal. We write,

$$L_i = Q_i, \quad (i = 1, \dots, n)$$

Substituting the expressions for L_i, Q_i into the above equation we obtain:

$$\frac{d}{dt} \frac{\partial T}{\partial \dot{q}_i} - \frac{\partial T}{\partial q_i} = Q_i, \quad (i = 1, \dots, n) \quad (3.5)$$

Equations (3.5) are known as the Lagrangian equations of motion of the second type. The kinetic energy of the system T , present in Equation (3.5), can be determined in terms of the generalized coordinates and time. Namely,

$$T = \frac{1}{2} \sum_{j=1}^N m_j \vec{v}_j \cdot \vec{v}_j = \frac{1}{2} \sum_{j=1}^N m_j \frac{d\vec{r}_j}{dt} \cdot \frac{d\vec{r}_j}{dt}$$

but

$$\frac{d\vec{r}_j}{dt} = \frac{\partial \vec{r}_j}{\partial t} + \sum_{i=1}^n \frac{\partial \vec{r}_j}{\partial q_i} \dot{q}_i$$

and therefore

$$T = \frac{1}{2} \sum_{j=1}^N m_j \left(\frac{\partial \vec{r}_j}{\partial t} + \sum_{i=1}^n \frac{\partial \vec{r}_j}{\partial q_i} \dot{q}_i \right) \cdot \left(\frac{\partial \vec{r}_j}{\partial t} + \sum_{i=1}^n \frac{\partial \vec{r}_j}{\partial q_i} \dot{q}_i \right)$$

After the dot product of the two vectors in the brackets is calculated and the resulting terms are grouped, we can obtain the following expression for the kinetic energy:

$$T = \frac{1}{2} \sum_{i,k=1}^n a_{ik} \dot{q}_i \dot{q}_k + \sum_{i=1}^n a_i \dot{q}_i + a_0 \quad (3.6)$$

where coefficients a_{ik} , a_i , a_0 are the functions of generalized coordinates and time, given by the following relations:

$$a_{ik} = \sum_{j=1}^N m_j \frac{\partial \vec{r}_j}{\partial q_i} \cdot \frac{\partial \vec{r}_j}{\partial q_k} \quad (3.7a)$$

$$a_i = \sum_{j=1}^N m_j \frac{\partial \vec{r}_j}{\partial q_i} \cdot \frac{\partial \vec{r}_j}{\partial t} \quad (3.7b)$$

$$a_0 = \sum_{j=1}^N m_j \frac{\partial \vec{r}_j}{\partial t} \cdot \frac{\partial \vec{r}_j}{\partial t} \quad (3.7c)$$

For the scleronomic type systems, such as essentially all multi-body systems with one-sided constraints, the time does not enter explicitly into the relations for the positional vectors $\vec{r}_j$. Henceforth,

$$\frac{\partial \vec{r}_j}{\partial t} = \vec{0} \quad , \quad (j = 1, \dots, N)$$

and thus the terms in Equation (3.7b) and Equation (3.7c) will vanish. The kinetic energy for such systems will therefore be given by:

$$T = \frac{1}{2} \sum_{i,k=1}^n a_{ik} \dot{q}_i \dot{q}_k \quad (3.8)$$

which in matrix form could be written as

$$T = \frac{1}{2} [q]^T \cdot [A] \cdot [q] \quad (3.9)$$

It is evident from Equation (3.7a) that matrix $[A]$ is symmetric. Hence,

$$\frac{\partial T}{\partial \dot{q}_\alpha} = \sum_{i=1}^n a_{\alpha i} \dot{q}_i \quad , \quad \frac{\partial T}{\partial q_\alpha} = \sum_{i,k=1}^n \frac{\partial a_{ik}}{\partial q_\alpha} \dot{q}_i \dot{q}_k$$

Substituting the above expressions into Equation (3.5) and remembering that for scleronomic systems,

$$\frac{d}{dt} = \sum_{i=1}^n \frac{\partial}{\partial q_i} \dot{q}_i$$

we can obtain the equation of motion for the system in terms of generalized coordinates:

$$\sum_{k=1}^n a_{\alpha k} \ddot{q}_k + \sum_{k=1}^n b_{\alpha k} \dot{q}_k = Q_{\alpha} \quad (3.10)$$

where:

$$b_{\alpha k} = \sum_{i=1}^n \left[\frac{\partial a_{\alpha k}}{\partial q_i} \dot{q}_i + \frac{1}{2} \frac{\partial a_{ik}}{\partial q_{\alpha}} \dot{q}_i \right]$$

In matrix form Equation (3.10) can be written as

$$[A] \cdot [\ddot{q}] + [B] \cdot [\dot{q}] = [Q] \quad (3.11)$$

The above system of equations represents the Lagrangian equations of motion for a scleronomic system of N particles in terms of the generalized coordinates.

It was assumed that $\{q_i\}$ is a set of independent coordinates. If such is the case than it can be shown that matrix $[A]$ in Equation (3.11) achieves a full rank, i.e., is non-singular, and the equations can be solved for the generalized coordinates uniquely. If, however, the generalized coordinates are dependent then the matrix $[A]$ will not achieve the full rank, and will become singular. This can occur if, for example, four or more generalized coordinates are used to describe the three Cartesian coordinates of the position of one of the particles.

A situation may arise such that the number of the generalized coordinates chosen is higher then the number of

degrees of freedom of the system. This may not necessarily result in the singularity of the matrix, and the system of equations may be solvable. In such a case the solution does not reflect all the constraints that the system was originally subjected to.

On the other hand, during the motion additional constraints may suddenly be imposed upon the system. These may frequently occur in systems for which the topology does not remain constant. Each additional constraint reduces the number of degrees of freedom of the system and creates a dependence among the generalized coordinates. Consequently, there are more generalized coordinates than are needed to describe the system, and hence some of them become redundant. Clearly, the coordinate dependencies resulting from these constraints, have to be included in the system analysis, otherwise the solution of the equations of motion will not represent the motion of a constrained system. This can be achieved through what is known as the constraint equations, which are discussed next.

3.3 CONSTRAINT EQUATIONS

The constraint equations are equations indicating the geometrical and kinetic dependencies among the particles comprising a given system. In the present analysis only the geometrical constraints are considered, as only such will be encountered in systems for which our model is designed.

The geometrical constraints establish the spatial dependencies among the system's particles, and, for scleronomic systems, can be described by the following equation:

$$f_{\alpha}(\vec{r}_j) = 0, \quad (\alpha = 1, \dots, d)$$

where d indicates the number of dependent (redundant) coordinates in the chosen system of coordinates. As an example let us consider a geometrical constraint defining a fixed distance L between two particles i and j in the system. The corresponding constraint equation would be given by:

$$(\vec{r}_i - \vec{r}_j) \cdot (\vec{r}_i - \vec{r}_j) - L^2 = 0$$

The constraint equations can be written in terms of the generalized coordinates,

$$f_a(q_1, q_2, \dots, q_n) = 0 \quad (3.12)$$

There are two basic methods to handle the constraints. The first method, which suggests itself naturally, is based on solving the additional constraint equations (which are here assumed to be holonomic or integrable) for the redundant coordinates explicitly in terms of the remaining coordinates (now independent), and then substituting them into the equations of motion (Equation (3.11)). This approach is not very feasible especially for large systems as it will greatly increase the complexity of already complicated equations of motion. The second method makes use of Lagrange multipliers, and provides much more convenient means of handling the constraint equations.

In a moving system of particles, constraints result in reaction forces. We recall that the equations of motion which were derived from the virtual work principle, Equation (3.1), carried the assumption that the virtual work of the system's reaction forces was zero. This is of course true, since in the properly defined (in terms of generalized coordinates) system the relative virtual displacements are possible only in the directions normal to the corresponding reaction (constraint) forces. However, if the chosen system of coordinates contains the redundant coordinates then the virtual displacements will not be in conformity with the

constraints. Hence, the virtual work by the reaction forces in the constraints will not be zero, and thus must be included.

In terms of the generalized coordinates, the virtual displacement in a constraint, by Equation (3.12), is:

$$\delta f_\alpha = \sum_{i=1}^n \frac{\partial f_\alpha}{\partial q_i} \delta q_i \quad (3.13)$$

The virtual work by the constraint forces would then be:

$$\delta W_f = \sum_{\alpha=1}^d \lambda_\alpha \delta f_\alpha = \sum_{\alpha=1}^d \left[\lambda_\alpha \sum_{i=1}^n \frac{\partial f_\alpha}{\partial q_i} \delta q_i \right] \quad (3.14)$$

where λ_α are constraint forces in a generalized sense, called Lagrange multipliers. Similarly to Equation (3.3), we can change the order of summation and write Equation (3.14) as

$$\delta W_f = \sum_{i=1}^n \Lambda_i \delta q_i, \quad \Lambda_i = \sum_{\alpha=1}^d \lambda_\alpha \frac{\partial f_\alpha}{\partial q_i} \quad (3.15)$$

The above expression represents the virtual work done by the unknown constraint forces and thus must be added to the LHS of Equation (3.4). Hence, the equations of motion (Equation (3.5)) take on the following form:

$$\frac{d}{dt} \frac{\partial T}{\partial \dot{q}_i} - \frac{\partial T}{\partial q_i} = Q_i + \Lambda_i, \quad (i = 1, \dots, n) \quad (3.16)$$

By the analogy to Equation (3.11), the equations of motion can be written in a matrix form. Namely,

$$[A] \cdot [\ddot{q}] + [B] \cdot [\dot{q}] = [Q] + [\Lambda] \quad (3.17)$$

where:

$$[\Lambda] = [G]^T \cdot [\lambda], \quad g_{\alpha i} = \frac{\partial f_{\alpha}}{\partial q_i}$$

Equation (3.17) represents a system of n differential equations in $n+d$ unknowns $\{q_i\}$, $\{\lambda_{\alpha}\}$, which together with the set of d algebraic constraint equations (Equation (3.12)) comprise a system of differential algebraic equations (DAE) of motion of the system.

Instead of solving the constraint equations for the redundant coordinates, the above system of DAEs can be solved. The solution of these equations for the generalized coordinates will reflect the constrained motion of the system.

3.4 TREATMENT OF RIGID BODIES

The equations of motion derived in the previous two sections can be readily applied to a system containing rigid bodies instead of particles. Indeed, the virtual work principle, Equation (3.1), will now contain the rotational terms, in addition to the linear terms. We write:

$$\sum_{j=1}^N [(\vec{F}_j - m_j \vec{a}_j) \cdot \delta \vec{r}_j + (\vec{M}_j - I_j \vec{\alpha}_j) \cdot \delta \vec{\theta}_j] = 0 \quad (3.18)$$

where $\vec{M}_j$ is the moment about a certain point in the body from all the external loads applied, I_j is the body's moment of inertia about the instantaneous axis of rotation passing through that point, $\vec{\alpha}_j$ is angular acceleration vector, $\delta \vec{\theta}_j$ is the vector of virtual angular displacements, and,

$$\vec{r}_j = \vec{r}_j(q_1, \dots, q_n)$$

$$\vec{\theta}_j = \vec{\theta}_j(q_1, \dots, q_n)$$

Repeating the procedure of Section 3.2 we can write the equations of motion for a system of rigid bodies, similarly to Equation (3.16),

$$\frac{d}{dt} \frac{\partial T}{\partial \dot{q}_i} - \frac{\partial T}{\partial q_i} = Q_i + \Lambda_i, \quad Q_i = \sum_{j=1}^N \left[\vec{F}_j \cdot \frac{\partial \vec{r}_j}{\partial q_i} + \vec{M}_j \cdot \frac{\partial \vec{\theta}_j}{\partial q_i} \right] \quad (3.19)$$

where T is the kinetic energy of the system and Λ_j is given by Equation (3.15).

In general, the kinetic energy of a rigid body is given by (see Appendix B),

$$T_j = \frac{m_j}{2} \left(\frac{d\vec{r}_{0j}}{dt} \right)^2 + m_j \frac{d\vec{r}_{0j}}{dt} \cdot \frac{d\vec{r}_{c.g.j}}{dt} + \frac{I_j}{2} \left(\frac{d\vec{\theta}_j}{dt} \right)^2 \quad (3.20)$$

where $\vec{r}_{0j}$ is the positional vector of some chosen point within rigid body j in the inertial frame of reference, $\vec{r}_{c.g.j}$ is the vector from that point to the body's center of gravity in the local frame of reference (fixed to the body), and $\vec{\theta}_j$ is the angular position vector of the local reference frame. The kinetic energy of the system is now given by:

$$T = \sum_{j=1}^N T_j = T_1 + T_2 + T_3 \quad (3.21)$$

where T_1 , T_2 , T_3 correspond to the first, second, and third terms in Equation (3.20). By the analogy to Section 3.2 terms T_1 , T_3 , can be evaluated according to Equation (3.6) in terms of generalized coordinates,

$$T_1 = \frac{1}{2} \sum_{i,k=1}^n a_{ik}^1 \dot{q}_i \dot{q}_k, \quad a_{ik}^1 = \sum_{j=1}^N m_j \frac{\partial \vec{r}_{0j}}{\partial q_i} \cdot \frac{\partial \vec{r}_{0j}}{\partial q_k} \quad (3.22a)$$

$$T_3 = \frac{1}{2} \sum_{i,k=1}^n a_{ik}^3 \dot{q}_i \dot{q}_k, \quad a_{ik}^3 = \sum_{j=1}^N I_j \frac{\partial \theta_j}{\partial q_i} \cdot \frac{\partial \theta_j}{\partial q_k} \quad (3.22b)$$

Evaluation of term T_2 is more complicated as it involves a triple product. We have, for a single body,

$$T_{2j} = m_j \frac{d\vec{r}_{0j}}{dt} \cdot \frac{d\vec{r}_{c.g.j}}{dt} = m_j \frac{d\vec{r}_{0j}}{dt} \cdot \frac{d\theta_j}{dt} \times \vec{r}_{c.g.j}$$

which in terms of the generalized coordinates can be written as:

$$T_{2j} = m_j \sum_{i,k=1}^n \left(\sum_{p,r,s} \epsilon_{prs} \frac{\partial r_{0j}^s}{\partial q_i} \frac{\partial \theta_j^r}{\partial q_k} r_{c.g.j}^p \right) \dot{q}_i \dot{q}_k$$

where superscripts p, r, s denote the cartesian coordinates of the respective vectors and ϵ_{prs} is the permutation constant. Summing over all the rigid bodies we obtain the following relation:

$$T_2 = \frac{1}{2} \sum_{i,k=1}^n a_{ik}^2 \dot{q}_i \dot{q}_k \quad (3.22c)$$

where

$$a_{ik}^2 = 2 \sum_{j=1}^N m_j \left(\sum_{p,r,s} \epsilon_{prs} \frac{\partial r_{0j}^s}{\partial q_i} \frac{\partial \theta_j^r}{\partial q_k} r_{c.g.j}^p \right)$$

Finally,

$$T = \frac{1}{2} \sum_{i,k=1}^n a_{ik} \dot{q}_i \dot{q}_k, \quad a_{ik} = a_{ik}^1 + a_{ik}^2 + a_{ik}^3 \quad (3.23)$$

The generalized forces Q_i and the kinetic energy T of the system were thus determined in terms of the generalized coordinates q_i . The final form of the differential algebraic equations (DAEs) of motion for a system of N rigid bodies can be written as:

$$\left. \begin{aligned} [A] \cdot [\ddot{q}] + [B] \cdot [\dot{q}] &= [Q] + [G]^T \cdot [\lambda] \\ f_\alpha &= 0, \quad (\alpha = 1, \dots, d) \end{aligned} \right\} \quad (3.24)$$

where the coefficients a_{ik} of matrix $[A]$ are defined by Equation (3.23), coefficients b_{ik} of matrix $[B]$ are related to a_{ik} through Equation (3.10), the generalized forces Q_i are given by Equation (3.19), and the coefficients $g_{\alpha i}$ of matrix $[G]$ are established in Equation (3.17). Provided that the positional and orientational vectors $\vec{r}_j, \vec{\theta}_j$ for every rigid body in the system are known, these coefficients can always be evaluated and the system of equations (3.24) set up.

The generation of the system of DAEs of Equation (3.24) on a digital computer for an arbitrary system of rigid bodies was developed in [31], and the appropriate algorithms can be found in that reference. As a final note to the presentation of the mathematical model, a proposed system of coordinates used in the present model should be discussed. This will be done briefly in the next section.

3.5 PROPOSED SYSTEM OF GENERALIZED COORDINATES

In the previous section the Lagrangian equations of motion for a general three-dimensional case were established. In this section we will present our choice of generalized coordinates. Herein we shall limit ourselves only to a two-dimensional case, since at the present our model is only two-dimensional.

We recall that rigid bodies are approximated by disks. Simple rigid bodies, which are represented by single disks, are treated like particles, and thus can have a maximum of two degrees of freedom, that is their rotational motion is neglected. On the other hand, the complex rigid bodies, which are represented by rigid configurations of disks will have a maximum of three degrees of freedom (one rotational and two translational).

For a single isolated disk the chosen generalized coordinates represent the cartesian coordinates of the center of the disk. Hence,

$$\begin{aligned} q_1 &= x_j \\ q_2 &= y_j \end{aligned} \quad \rightarrow \quad \vec{r}_j = q_1 \mathbf{i} + q_2 \mathbf{j}$$

A single disk being in contact with another object in the system has only one degree of freedom. Consequently, its position is represented by only one generalized coordinate, provided that the object's location in the system is known. Figure 3.1 shows the choices of the generalized coordinates in cases when the disk is contacting an obstruction line segment (a), or another disk (b).

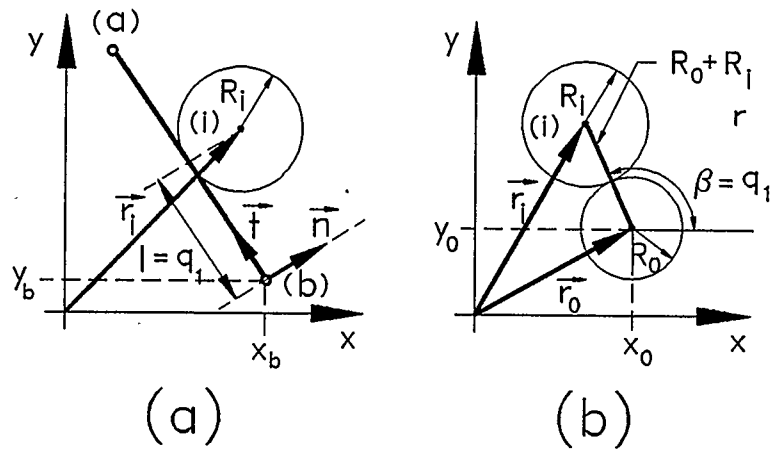


Figure 3.1. Definition of chosen generalized coordinates for a disk in contact with a line segment (a) and a disk (b).

For a disk in contact with an obstruction line segment, the generalized coordinate is chosen as the distance from one of the ends of the line segment to the point of contact. We thus have:

$$q_1 = l$$

$$\downarrow$$

$$\vec{r}_j = (x_b + q_1 t_x + R_j n_x) \mathbf{i} + (y_b + q_1 t_y + R_j n_y) \mathbf{j}$$

where n_x, n_y, t_x, t_y are cartesian components of the normal and tangential vectors for the given line segment, as shown in the figure.

For a single disk contacting another disk, the generalized coordinate is chosen as the angle that the line connecting the centers of the two disks makes with the x-axis, Figure 3.1b. We therefore have:

$$q_1 = \beta$$

↓

$$\vec{r}_j = (x_0 + (R_j + R_0) \cos(q_1)) \mathbf{i} + (y_0 + (R_j + R_0) \sin(q_1)) \mathbf{j}$$

where x_0, y_0 are the components of the known positional vector of the other disk.

Figure 3.2 shows the definition of the generalized coordinates for a complex rigid body.

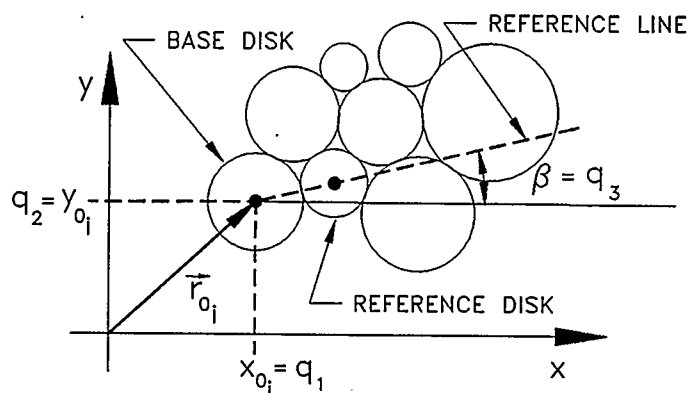


Figure 3.2. Definition of generalized coordinates for a complex rigid body.

The chosen generalized coordinates represent the two cartesian components of the positional vector of the base disk, and the angle that the reference line makes with the x-axis. The reference line is defined as a line connecting the centers of the base disk and some other disk selected within the body. This line defines the angular position of the entire body. The positional and angular vectors can be defined as:

$$\begin{aligned}
 q_1 &= x_{0j} , & q_2 &= y_{0j} , & q_3 &= \beta \\
 & & & \Downarrow & & \\
 \vec{r}_{0j} &= q_1 \mathbf{i} + q_2 \mathbf{j} , & \vec{\theta}_j &= q_3 \mathbf{k}
 \end{aligned}$$

If the given complex body is in contact with another object in the system then the contacting disk is always chosen as the base disk for that body. Thereafter, the generalized coordinates describing the positional vector for this disk ($\vec{r}_{0j}$) are chosen in the same way as for a single disk in contact.

This concludes the mathematical formulation of the present model. The positional and angular vectors for every body in the system are defined. The kinetic energy as well as the generalized forces can be determined in terms of the proposed generalized coordinates.

3.6 CONCLUSIONS

In this chapter the mathematical formulation for the model was presented. The motion of the system was described in terms of a set of simultaneous differential algebraic equations (DAE). Equation (3.24) shows the final form of the equations of motion. It should be pointed out here that the mathematical formulation of the problem was exact, as no simplifying assumptions were made during the formulation. Any such assumptions that are later imposed, are done so for the purpose of reducing the difficulty in obtaining a numerical solution and increasing the computational speed, or for the purpose of reducing the number of generalized coordinates. Treating a single disk as a particle is an example of the latter. Also, inadvertently, for many systems some simplifications will have to be made in the determination of the character of the external loads acting on the system which will lead to inherent errors in the generalized forces. Nevertheless, the mathematical formulation for the model is exact.

CHAPTER 4

GENERATION OF RIGID BODIES

4.1 INTRODUCTION

The processes involving the systems of a large number of rigid bodies nearly always possess a certain degree of randomness. This randomness is associated with a large variety of shapes and sizes of the rigid bodies as well as their varying concentrations and spatial distributions throughout the system. In computer simulations of processes such as these it is useful to be able to generate the rigid bodies at random, thus avoiding a biases associated with systems for which the rigid bodies are predefined. For this purpose a random number generator was incorporated into the computer model. This generator provides our model with a uniformly distributed sequence of random numbers which are, consequently, sampled to randomly generate the sizes of rigid bodies and their spatial positions according to specified distributions. The last statement seems at first sight to be contradictory. However, as we shall see in Section 4.3, it is possible to generate a given variable associated with a rigid body (such as its size, for example) completely at random in such a way that globally, this variable will be distributed according to some specified function.

Before presenting the methodology of rigid bodies generation and its numerical implementation, several basic ideas need to be discussed. Let us consider the example shown in Figure 4.1.

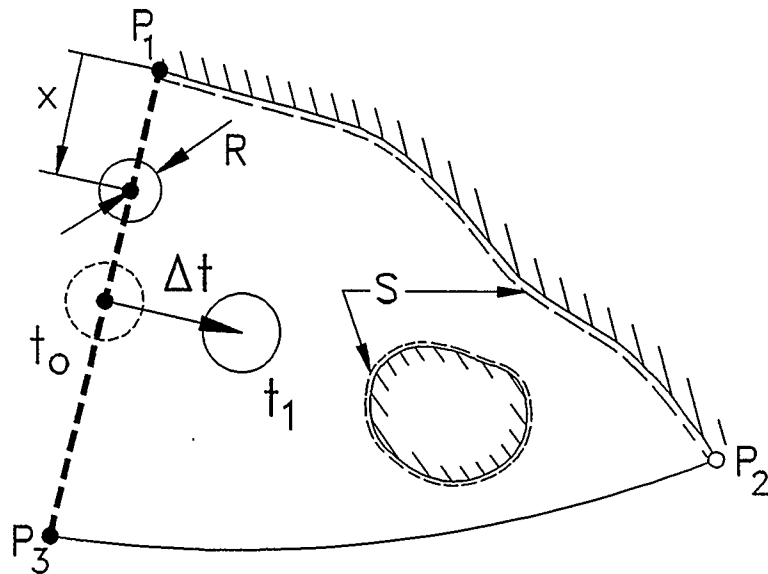


Figure 4.1. A schematic diagram for the rigid body generation.

The broken line in the figure represents a conveniently chosen boundary of the control area or the domain in which the simulation is carried out. The three main types of boundaries that can be identified are:

a) Solid Boundary, indicated by the line $(P_1 P_2)$ as well as by the closed curve S . This type of boundary typically represents physical structures such as the walls of a channel

or an obstruction.

b) Generation Boundary, represented in the figure by the line segment $(P_1 P_3)$. This boundary is chosen by the user and is used for the rigid body generation.

c) Exit Boundary, indicated by the segment $(P_3 P_2)$. Once a rigid body generated on the generation boundary crosses the exit boundary it is automatically deleted from the simulation procedure.

In our simulation program the disks are generated at the generation boundary one at a time. They move towards the solid boundary being acted on by the external forces of the system. Consequently, as the simulation progresses, more and more disks are present in the system. Every disk generation is considered as a random event, characterised by three random variables associated with it. Those are the size of a disk represented by its radius R , its position along the generation boundary x , and the time interval Δt after which the next generation is taking place. The disk generation procedure thus is reduced to generating the above three random variables. These variables are assumed to be continuous over their respective domains, for example, the radius of a disk can vary continuously from some minimum to some maximum value. Each of the random variables has a certain probability distribution function. If X denotes a random variable and x is its value then, the cumulative distribution function $F(x)$

is defined as the probability that a random selection of X gives a value less than x . We write

$$F_X(x) = P\{X < x\} \quad (4.1)$$

From the definition of the distribution function it is evident that $0 \leq F_X(x) \leq 1$. If a given random variable X has all values fall within some interval, $x_{\min} \leq x \leq x_{\max}$, then

$$F_X(x_{\min}) = 0, \quad F_X(x_{\max}) = 1$$

We can also identify the probability that the value of a random variable X falls within a specified interval (x_1, x_2) . Namely, if $(x_1, x_2) \in (x_{\min}, x_{\max})$ then

$$P\{x_1 \leq X \leq x_2\} = F_X(x_2) - F_X(x_1) \quad (4.2)$$

In the specific case when the size of the interval is (dx) we may write Equation (4.2) as

$$P\{X \in dx\} = F_X(x+dx) - F_X(x) = f_X(x) dx \quad (4.3)$$

where $f_X(x) = dF_X(x) / dx$ is known as the probability density function of a random variable X .

Conversely, if a probability density function for a given

random variable is known then the cumulative distribution can be calculated:

$$F_X(x) = \int_{x_{\min}}^x f_X(\eta) d\eta \quad (4.4)$$

The above result suggests that the cumulative distribution of a random variable is equal to the area under the probability density function, as shown in Figure 4.2.

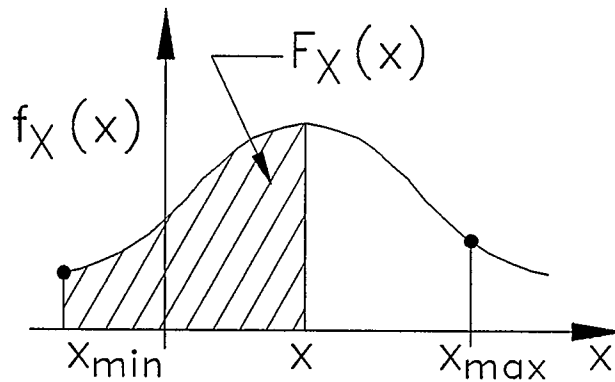


Figure 4.2. Correlation between $F(x)$ and $f(x)$.

Each of the three random variables associated with the generation of a rigid body has a probability density function associated with it as well as a uniformly distributed random number sequence. The sequence is used to randomly sample the given variable according to its distribution.

The methods of obtaining uniformly distributed random numbers on the $(0,1)$ interval are presented in Section 4.2 and

the sampling techniques are discussed in Section 4.3.

For the purpose of this model it is assumed that the radii of the disks are distributed according to a Gaussian distribution, their positions along the generation boundary are described by a uniform distribution, and the time increments are Poissonian inputs. The methodology of sampling of these random variables is presented in detail in Section 4.4.

4.2 RANDOM NUMBER GENERATORS

With the recent evolution in digital computers and their improved capabilities there is an increased interest in computational methods (such as Monte Carlo methods or stochastic simulation), in which certain phenomena or processes are studied by carrying out the computations for a large number of sets of randomly chosen parameters of the processes. Such methods make use of random numbers in modelling the process or computing an outcome of an event which is stochastic in nature. By stochastic process it is meant a process described by a sequence of states whose evolution is determined by random events. During the computation such random events are prescribed by the random numbers used.

Many physical devices have been constructed for the purpose of generating random numbers. Those are based on physical processes which are statistically considered random. There are also random number tables available on disks and magnetic tapes, which were generated by the physical processes and subjected to a number of statistical tests (references to the rand tables can be found in [15]). These tables are often considered a standard input for the computer programs which rely on the randomness in the strictest sense.

Often in practice an extremely high degree of randomness of the numbers may not be required for simulation of a particular phenomenon. In such instances it is convenient to generate the numbers one at a time, as they are needed, by a specified rule. The rule can be devised in such a way that the generated sequence of numbers will have desired statistical properties and no significant deviation from randomness will be detected when the sequence is subjected to statistical tests. Such a sequence of numbers is called pseudo-random and, although, not genuinely random, it can be close enough to randomness for practical applications. The advantage of using computer generated random numbers in the simulation of stochastic processes is that any given sequence of numbers can be readily reproduced if, for example, a verification of results is required or a comparison of different methodologies is made.

Most of the pseudo-random number (PRN) generators are based on recurrence formulas where the next number in the sequence is obtained from its predecessor. Different types of PRN generators are discussed in Appendix C.

For the purpose of our model a mixed congruential generator (Equation (C.8)) is chosen, based on [21].

$$x_{n+1} = \lambda x_n + b \pmod{P}$$

At the present time the simulation is carried out on a 32-bit binary machine. Therefore, 2^{32} is selected as the base of number representation P . The parameter λ is chosen so it satisfies the conditions necessary for the generator to attain a maximum period of $\tau_{\max} = 2^{32}$ (see Appendix C). Reference [21] suggests 1812433253 as the value for the parameter. The increment b should not have a common divisor with P and hence, for our choice of P any odd integer can be used. Parameter b is selected to be equal to the prime number 317.

In summary the PRN generator used for obtaining the PRN sequences ξ_i on $(0,1)$ interval, is in our model as follows,

$$\begin{aligned} x_{n+1} &= \lambda x_n + b \pmod{P} \\ \xi_{n+1} &= \frac{x_{n+1}}{P} \end{aligned} \tag{4.5}$$

with $P = 2^{32}$

$$\lambda = 1812433253 \quad [\equiv 1 \pmod{P}]$$

$$b = 317$$

x_0 a large odd number.

Our computer model requires three separate PRN sequences. Those are obtained by starting the generator of Equation (4.5) with three different initial numbers or seeds, x_0 . Also, these numbers are used by the program only at the time of the rigid body generation. There is, consequently, no need to store the entire sequence. The routine for generating random

numbers is thus designed in such a way that the new number is stored in place of its predecessor. Such a generator automatically reseeds itself. Logically this is written as

$$x \leftarrow \lambda x + b \pmod{P} \quad (4.6)$$

If numbers ξ_i are drawn at random from the $(0,1)$ interval, then their distribution should be uniform over the interval. This is evident since "randomness" implies that the probability of choosing any particular number from this interval is the same. A good PRN generator will supply a sequence of numbers with a uniform distribution. Hence, we write

$$f_{\xi}(\xi) = 1 \quad (4.7)$$

and the cumulative distribution by Equation (4.4) becomes,

$$F_{\xi}(\xi) = \xi, \quad \xi \in (0,1) \quad (4.8)$$

Now, by Equation (4.2) the probability that a chosen random number falls within a specified interval, $\xi \in (\xi_1, \xi_2)$, is

$$P\{\xi \in (\xi_1, \xi_2)\} = \xi_2 - \xi_1 \quad (4.9)$$

The selected PRN generator can also be used for the generation of pairs of random numbers (ξ, η) with a uniform distribution over a square with a unit side. A uniform

distribution of pairs of numbers implies that

$$P\{(\xi, \eta) \in \Delta A\} = \Delta A \quad (4.10)$$

where $\Delta A \in \mathbb{R}^2$ is some area within the square.

4.3 SAMPLING TECHNIQUES

In the computer simulation of stochastic processes, as is the case in the present study, it is often required that certain variables be drawn randomly from a specified distribution, in other words, that randomly chosen variables be distributed according to a given distribution function. Such selection of random variables is called sampling.

To define sampling mathematically, let us consider some space Γ_0 which can be either discrete or continuous. We denote the elements of this space as $x, x \in \Gamma_0$. We assume that there exists a continuous or discrete probability density function, $f(x)$ such that:

$$\int_{\Gamma_0} f(x) dx = 1 \quad (4.11)$$

Sampling is now defined as a procedure of producing a sequence of random variables $x_1, x_2, \dots, x_i, \dots$, such that for any sub-space $\Gamma \subset \Gamma_0$,

$$P\{x_i \in \Gamma\} = \int_{\Gamma} f(x) dx \leq 1. \quad (4.12)$$

We note that the proper rules of integration should be obeyed when a discrete space or a discrete distribution function is used, although the above equations are written for both cases.

If a given space is one-dimensional, such as an interval $(x_{\min}, x_{\max}) \in \mathbb{R}$ then,

$$P\{x_i \in (a, b)\} = \int_a^b f(x) dx \quad (4.13)$$

for any sub-interval $(a, b) \in (x_{\min}, x_{\max})$. In the limiting case when the given interval becomes an infinitesimal neighbourhood δx of point x then we may write

$$P\{x_i \in \delta x\} = f(x) |\delta x| \quad (4.14)$$

The present analysis will be limited to sampling one-dimensional distributions as only such are used in the model. One can, however, develop similar sampling techniques for multi-dimensional distributions.

We start with a set of random numbers, ξ_i , uniformly distributed on $(0,1)$. It is assumed that the numbers are provided by the generator of Equation (4.5) and have the cumulative distribution given by Equation (4.8). Sampling of a random variable $x \in (x_{\min}, x_{\max})$ with the corresponding probability density function $f(x)$ can consequently be reduced to finding a transformation function $\mathcal{F}$ which maps (one to one) the $(0,1)$ interval onto the $(x_{\min}, x_{\max})$ interval,

$$\mathcal{F} : \xi \rightarrow x \quad \begin{array}{l} \xi \in (0, 1) \\ x \in (x_{\min}, x_{\max}) \end{array} \quad (4.15)$$

so that a set of uniformly distributed numbers $\{\xi_i\}$ produces a set of random variables $\{x_i\}$ which are distributed according to $f(x)$. If we also require that the transformation function be non-decreasing then:

$$P\{\xi_i < \xi\} = P\{\mathcal{F}(\xi_i) < \mathcal{F}(\xi)\} \quad (4.16)$$

and by Equation (4.15)

$$P\{\xi_i < \xi\} = P\{x_i < x\} \quad (4.17)$$

Hence, from Equation (4.1), we have

$$F_\xi(\xi) = F_x(x) \quad (4.18)$$

and finally with the application of Equation (4.4) and Equation (4.8) we may write,

$$\xi = F_x(x) = \int_{x_{\min}}^x f(\eta) d\eta \quad (4.19)$$

The desired transformation function $\mathcal{F}$ can now be defined by solving the integral equation (4.19) in terms of x . This is equivalent to finding an inverse of the cumulative distribution function, denoted here as F_x^{-1} . In short we can write:

$$x = \mathcal{F}(\xi) = F_x^{-1}(\xi) \quad (4.20)$$

One could anticipate the result of Equation (4.19). A similarity between ξ and $F(x)$ suggests itself since both have their values between 0 and 1. If we again consider a small interval δx , then from Equation (4.3), the LHS of Equation (4.14) is simply $\delta F_X(x)$. $\delta F_X(x)$ represents a probability that the value of the random variable x falls within δx . Now, if for any such interval δx we choose a corresponding infinitesimal interval $\delta \xi = \delta F_X(x)$ and, moreover, if we assume that for any random number $\xi_i \in \delta \xi$ we select a random variable such that $x_i \in \delta x$, then

$$P\{x \in \delta x\} = P\{\xi \in \delta \xi\} = \delta \xi = f(x) \delta x \quad (4.21)$$

The above result is analogous to Equation (4.14). Hence, the random variable x is distributed according to $f(x)$. Such a mapping of infinitesimal intervals is equivalent to the transformation function of Equation (4.15) since $d\xi = f(x) dx$ represents relation (4.19). This concludes our argument. Figure 4.3 shows the graphical representation of the mapping which defines the sampling procedure.

The sampling procedure described above is often referred to as sampling by a variable transformation. We generate a sequence of random numbers $\{\xi_i\} \in (0, 1)$, and then apply a transformation of Equation (4.20) to obtain the desired sequence of random variables $\{x_i\} \in (x_{\min}, x_{\max})$. Often one may

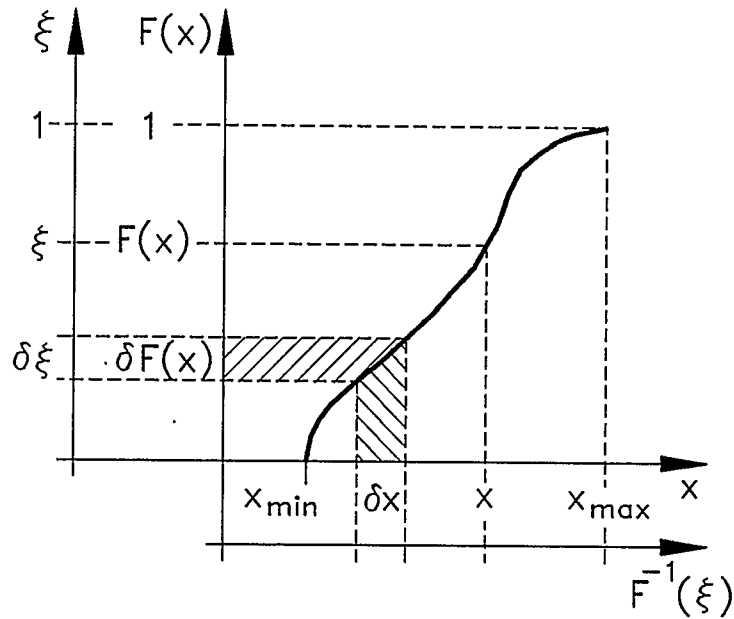


Figure 4.3. A graphical representation of the mapping of $(0,1)$ onto x -domain.

not be able to find an analytical expression for the transformation function (F_x^{-1}) , and a numerical solution of the integral equation (4.19) may not be economical. If such is the case, sampling can be performed by what is known as discretization of the probability density function.

Let us consider a one-dimensional domain $(x_{\min}, x_{\max})$ with a continuous probability density function, $f(x)$, prescribed on it, as shown in Figure 4.4. We divide the interval $(x_{\min}, x_{\max})$ into a series of small finite rather than infinitesimal intervals $\Delta x_1, \Delta x_2, \dots, \Delta x_n$ over which the probability density function (PDF) can be assumed constant. If $f_1, f_2, \dots, f_n$ are the values of PDF over the respective

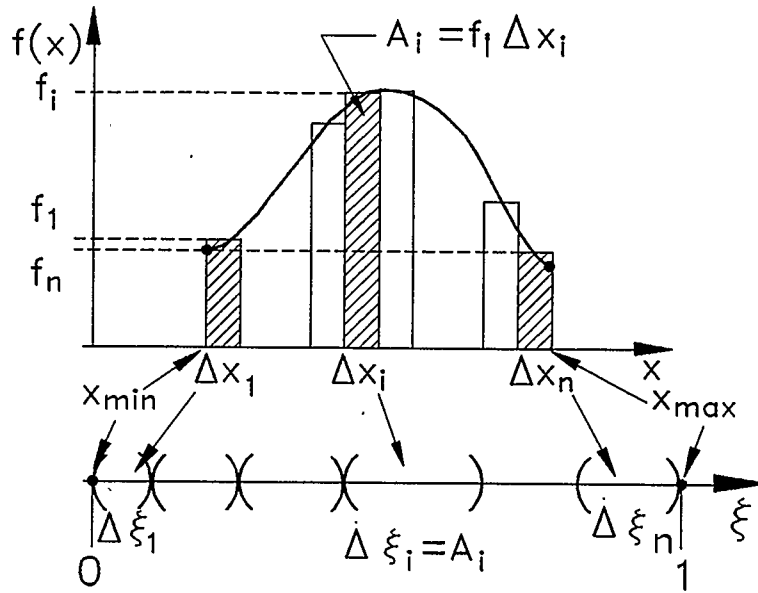


Figure 4.4. Discretization of the probability density function $f(x)$.

intervals, then

$$\sum_{i=1}^n f_i \Delta x_i = 1 \quad (4.22)$$

and

$$P\{x \in \Delta x_i\} = f_i \Delta x_i \quad (4.23)$$

The above result suggests that selecting a random variable $x \in \Delta x_i$ can be thought of as a random event the probability of which is equal to $f_i \cdot \Delta x_i$. We have n such intervals, and hence, sampling process reduces to randomly choosing one of the n random events $E_1, \dots, E_n$ with respective probabilities being $P\{E_i\} = f_i \cdot \Delta x_i$, $(i=1, n)$.

We now divide $(0,1)$ into n discrete subintervals $\Delta\xi_1, \Delta\xi_2, \dots, \Delta\xi_n$ and we say that if a generated random number falls within sub-interval i , i.e. $\xi \in \Delta\xi_i \in (0,1)$, then we select event E_i . From Equation (4.9) it can be seen that the probability that a random number on $(0,1)$ falls within a specified interval is simply equal to the length of this interval, i.e. $P\{\xi \in \Delta\xi_i\} = \Delta\xi_i$. Consequently, if we require that event E_i have a specified probability then the length of the corresponding interval $\Delta\xi_i$ must be equal to this probability. We can write this condition as

$$P\{E_i\} = P\{\xi \in \Delta\xi_i\} = \Delta\xi_i$$

Since event E_i represents selecting random variable x from interval Δx_i then,

$$P\{E_i\} = P\{x \in \Delta x_i\} = f_i \cdot \Delta x_i$$

and therefore:

$$\Delta\xi_i = f_i \cdot \Delta x_i \quad (4.24)$$

The result of Equation (4.24) represents a mapping of discrete intervals in ξ -domain onto discrete intervals in x -domain, and since $f(x) = \text{const}$ on each interval, such mapping is described by a linear transformation function. We write this as $\mathcal{Q} : \xi \rightarrow x$, $\xi \in \Delta\xi_i$, $x \in \Delta x_i$. If we use superscripts 1

and 2 to denote the end points of the respective intervals then similarly to Equation (4.15) we may write

$$\mathcal{Q} : \xi \rightarrow x \quad \begin{cases} \xi \in (\xi_i^1, \xi_i^2) \\ x \in (x_i^1, x_i^2) \end{cases}$$

and by Equation (4.19) we have

$$\xi - \xi_i^1 = f_i (x - x_i^1)$$

Whence, the selected random variable x is given by

$$x = \mathcal{Q}(\xi) = x_i^1 + \frac{1}{f_i} (\xi - \xi_i^1) \quad (4.25)$$

It is evident from Equation (4.24) that $\mathcal{Q}(\xi_i^2) = x_i^2$. Also,

$$\sum_{i=1}^n \Delta \xi_i = 1.$$

The above method of "allocation and selection" will, thus, produce a set of random variables $\{x\}$ distributed according to $f(x)$ from a sequence of uniformly distributed random numbers $\{\xi\}$. These are, of course, provided by the PRN generator, Section 4.2.

Perhaps the easiest way of discretizing the two domains

is by selecting $n-1$ points $x^1, x^2, \dots, x^{n-1}$ within the given interval $(x_{\min}, x_{\max})$ such that

$$x_{\min} < x^1 < x^2 < \dots < x^{n-1} < x_{\max}$$

Now, if we denote $x_{\min} = x^0$, $x_{\max} = x^n$, then we can define the sub-intervals Δx_i as

$$\overline{\Delta x_i} \triangleq (x^{i-1}, x^i), \quad (i=1, n) \quad (4.26)$$

and

$$\Delta x_i = x^i - x^{i-1}$$

Constants f_i are taken as the values of PDF at the mid-points of the respective intervals. Thus,

$$f_i = f\left(\frac{x^i + x^{i-1}}{2}\right)$$

Using the definition of (4.26) we define intervals $\Delta \xi_i$ in such a way that relation (4.24) is satisfied. Namely, we set $\xi^0 = 0$, and we let

$$\xi_i = \sum_{j=1}^i f_j \cdot \Delta x_j, \quad (i=1, n) \quad (4.27)$$

and if

$$\overline{\Delta \xi_i} \triangleq (\xi^i, \xi^{i-1}) \quad (4.28)$$

then

$$\Delta \xi_i = \xi^i - \xi^{i-1} = \sum_{j=1}^i (f_j \cdot \Delta x_j) - \sum_{j=1}^{i-1} (f_j \cdot \Delta x_j) = f_i \cdot \Delta x_i$$

This satisfies condition (4.24) and hence, ξ -domain is discretized properly. Now the sampling methodology can be summarized as follows. We generate a random number $\xi \in (0,1)$ and locate the sub-interval within which that number falls ($\Delta \xi_i$ as defined by Equation (4.28)). This establishes the corresponding sub-interval Δx_i from which the random variable x is drawn according to relation (4.25).

As a final note we will now consider what is known as rejection sampling method. This is often, in terms of the computer time involved, the quickest and the most efficient sampling technique. It is a very general method and it can be used to sample virtually any probability distribution function. It is particularly useful for sampling distributions for which the variable transformation method cannot be easily applied, and offers advantages over the discretization technique where considerable computer time can be lost on searching through the intervals and allocating the variables. However, this method requires a good square PRN generator, i.e., a generator which is capable of generating uniformly distributed pairs of random numbers. As before we will again confine ourselves to one-dimensional distributions only. Let us consider a situation shown in Figure 4.5.

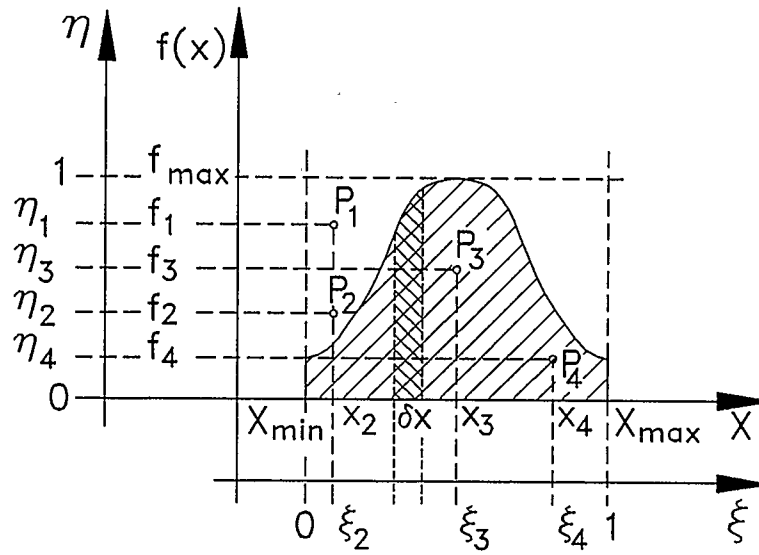


Figure 4.5. Sampling of the PDF by a rejection technique.

We wish to choose variables x at random from $(x_{\min}, x_{\max})$ in such a way that the variable is distributed according to $f(x)$. The sampling technique presented here relies on generating a pair of random numbers (ξ, η) , $0 \leq \xi, \eta \leq 1$. These two numbers correspond to coordinates of some point $P(x, f)$ in the x - f plane respectively, i.e.

$$\begin{aligned} \xi \in (0, 1) &\propto x \in (x_{\min}, x_{\max}) \\ \eta \in (0, 1) &\propto f \in (0, f_{\max}(x)) \end{aligned} \quad (4.29)$$

The linear correspondences in Equation (4.29) are given by,

$$\begin{bmatrix} \xi = \frac{x - x_{\min}}{x_{\max} - x_{\min}} \\ \eta = \frac{f}{f_{\max}(x)} \end{bmatrix} \Leftrightarrow \begin{bmatrix} x = x_{\min} + \xi \cdot (x_{\max} - x_{\min}) \\ f = \eta \cdot f_{\max}(x) \end{bmatrix} \quad (4.30)$$

where $f_{\max}(x) = f_{\max}$ denotes the maximum value of the PDF on the given interval $(x_{\min}, x_{\max})$. The probability density function consists of a set of points in x - f plane for which $f = f(x)$.

Equations (4.30) define a one-to-one transformation of ξ - η domain (a square with a unit side) into x - f domain represented by a rectangle with sides being $(x_{\min}, x_{\max})$ and $(0, f_{\max})$. Since the transformation is one-to-one, it is evident that every point (ξ, η) will have one, and only one, point (x, f) corresponding to it.

Sampling of x is now performed in the following way. We generate a pair of random numbers ξ, η . From Equation (4.30) we determine the corresponding point (x, f) . If this point falls below the PDF curve $f(x)$ (i.e., falls within the shaded area in the diagram in Fig.(4.5)) then we select variable x , otherwise we reject it and generate another pair of random numbers which, again, are tested, and so on. In the end we have a sequence of random variables $\{x\}$ which were accepted. The condition for accepting a variable x given a pair of random numbers ξ, η is that:

$$f \leq f(x) \quad \begin{cases} x = x_{\min} + \xi \cdot (x_{\max} - x_{\min}) \\ f = \eta \cdot f_{\max} \end{cases} \quad (4.31)$$

Lastly, we need to verify that the generated sequence of random variables has, indeed, a desired distribution $f(x)$. Let us consider an infinitesimal interval $\delta x \in (x_{\min}, x_{\max})$, as shown in Figure 4.5. We are interested in finding the probability that a selected random variable x falls within that interval. This is equivalent to estimating the probability that a generated point (x, f) belongs to the incremental area under the PDF curve δA provided that only successful points are considered. We can write this as:

$$P\{x \in \delta x \mid \text{success}\} = P\{(x, f) \in \delta A \mid \text{success}\}$$

which can be written as

$$P\{x \in \delta x \mid \text{success}\} = P\{x \in \delta x \mid f \leq f(x)\}$$

Since the variables x, f are obtained from a linear transformation of random numbers ξ, η , they are independent and uniformly distributed over their respective domains. Hence, the conditional probability on the LHS of the above equation is simply equal to the product of the probabilities of the components. We write:

$$P\{x \in \delta x \mid \text{success}\} = \delta x \cdot f(x)$$

The above result is the same as the one in Equation (4.14), and hence, we can conclude that random variables generated by this method will be distributed according to the given PDF. The method of selection and rejection presented above is relatively simple and can easily be implemented on a digital computer provided that a good square generator is available. The disadvantage is that for certain PDF's it may have a low efficiency, i.e., many values may be rejected before one is accepted in which case sampling by discretization may be a better technique. Such situation arises when the area under the PDF curve is small compared to the entire domain $f_{\max} \cdot (x_{\max} - x_{\min})$. Figure 4.6 shows a suitable and unsuitable PDFs for the application of rejection method.

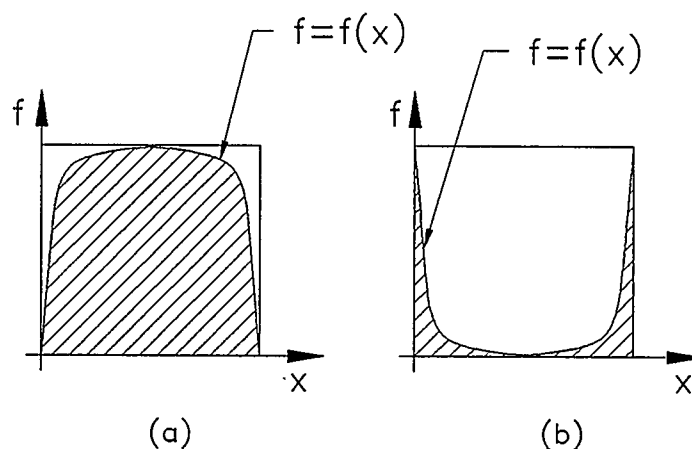


Figure 4.6. Typical PDF's, suitable (a) and unsuitable (b) for the application of rejection sampling technique.

4.4 METHODOLOGY OF GENERATING RIGID BODIES

In our model the generation of a rigid body is assumed to be a random event wherein a new body enters our control area somewhere on the generation boundary. It is evident that if a truly stochastic simulation of a process is to be carried out, then the point of entry must be random as must be the size of the rigid body and the time intervals between the consecutive entries (generations). For example, let us consider a process of interaction of ice carried by a river with a bridge pier. An observer standing on the river bank will see ice floes moving towards the bridge pier. It will seem to him that the floes entering his viewing area are of different sizes and shapes and that they appear with different frequencies and at different distances from the shore. He will not be able to establish any dependencies between the appearances of consecutive floes and will not be able to predict the spatial or physical characteristics of the next floe to enter the viewing area. In other words, an observer will perceive the appearance of a floe as a random event and the parameters associated with it as random variables. He may also notice that some sizes of the floes are more common than others as also may be the case with other variables. Hence, he may conclude that the characteristics of the floes, although random, may be distributed in a certain way, i.e.,

that certain values of the associated parameters may be more likely to occur than others.

In the simulation of such a process it is desirable to retain its stochastic nature. Hence, the generation of a rigid body (i.e., its entry into the control area) is interpreted as a random event. Each such event has three random variables associated with it, namely: size (radius of the disk), location at which it enters, and the time interval until the next generation (entry). Each random variable has a corresponding numerical value and an assumed distribution function. Consequently, the procedure of a rigid body generation boils down to random sampling of the three random variables. For this purpose we use the PRN generator of Section 4.2, which is capable of generating a sequence of random numbers on $(0,1)$ with good one- and two-dimensional distributions. Three independent PRN sequences, one for each variable, are started. Every time the rigid body generation is to take place, one random number from each sequence is generated. These are then used to sample the respective distribution functions for the values of the random variables. Sampling techniques were discussed in Section 4.3 and those will only be referenced in this section.

A schematic block diagram of the generation routine used by the model is shown in Figure 4.7. The methodologies for

sampling of particular random variables and their computer implementations are discussed in the consecutive subsections of this section.

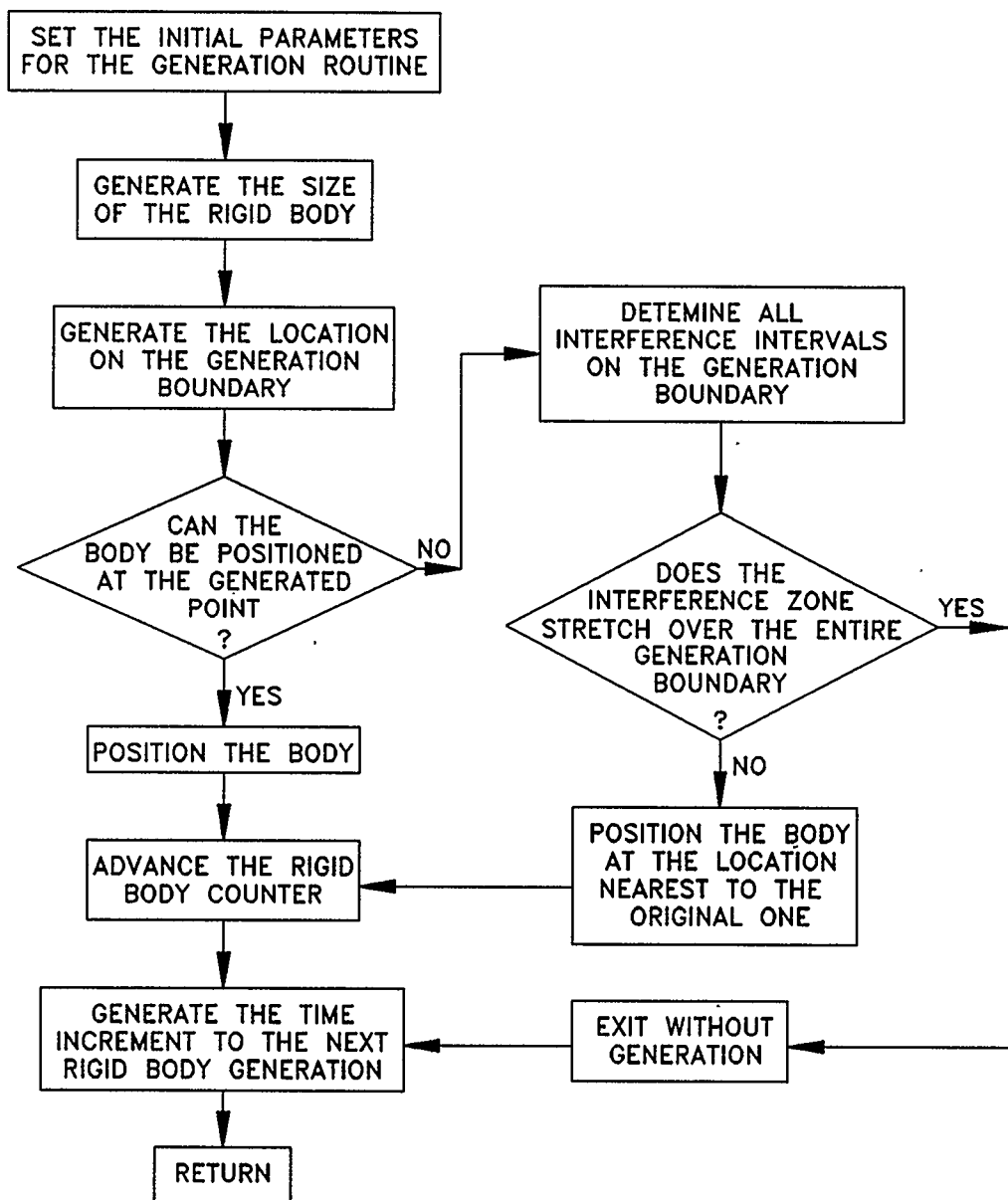


Figure 4.7. Block diagram of the generation routine

4.4.1 Sampling for the size of a rigid body

Rigid bodies are represented by disks whose sizes are determined by their radii. It is assumed that the radii are distributed according to a Gaussian distribution given by

$$f(x) = \frac{1}{\sigma\sqrt{2\pi}} \cdot \exp\left(-\frac{(x - \mu)^2}{2\sigma^2}\right), \quad (-\infty < x < \infty) \quad (4.32)$$

where μ is the expected or mean value,

$$\mu = E(x) = \int_{-\infty}^{+\infty} x f(x) dx$$

and σ^2 is the variance of the random variable defined as

$$\sigma^2 = \text{VAR}(x) = \int_{-\infty}^{+\infty} (x - \mu)^2 f(x) dx$$

The two constants, μ, σ , are the characteristic parameters of the distribution and must be entered as inputs into the program.

The cumulative distribution of the random variable X for the given PDF, by equations (4.1) and (4.4), is given by the following relation

$$F_X(x) = \int_{-\infty}^x f(y) dy = \int_{-\infty}^x \left[\frac{1}{\sigma\sqrt{2\pi}} \cdot \exp\left(-\frac{(y-\mu)^2}{2\sigma^2}\right) \right] dy = \text{erf}(x)$$

There is no analytical solution of the above integral and therefore sampling by a variable transformation cannot be used. One of the other two techniques is considered. Since the chosen PRN generator gives a good distribution for the pairs of numbers it is suggested that the selection-rejection sampling technique be used. It is also believed that the efficiency of the method is sufficient for a Gaussian PDF.

We will sample random variable X representing the radius of a disk by a rejection method (Section 4.3). The variable will be sampled from a finite interval $(x_{\min}, x_{\max})$ rather than an infinite one. The governing condition is shown in Equation (4.31). The maximum value of the PDF over the interval, provided, of course, that $\mu \in (x_{\min}, x_{\max})$, is

$$f_{\max} = f_{\max}(x) = f(x=\mu) = \frac{1}{\sigma\sqrt{2\pi}}$$

The points $x_{\min}, x_{\max}$ correspond, respectively, to a minimum and a maximum value for the radii of the disks used by the program and are entered as inputs.

Using the PRN generator we generate two random numbers ξ, η . We then apply the transformation equation (4.30) to

obtain the corresponding pair of variables, x, f , i.e.,

$$\begin{cases} x = x_{\min} + \xi \cdot (x_{\max} - x_{\min}) \\ f = \eta \cdot \frac{1}{\sigma \sqrt{2\pi}} \end{cases}$$

Now if

$$f \leq \frac{1}{\sigma \sqrt{2\pi}} \cdot \exp\left(-\frac{(x - \mu)^2}{2\sigma^2}\right)$$

then we accept the variable x . If not then we generate another pair of random numbers and repeat the above steps. This procedure is repeated until a successful selection is made. The generated size is then

$$R = x$$

This concludes the methodology of sampling for size. The appropriate routine was programmed on the computer.

4.4.2 Sampling for position

The position of a rigid body at the entrance into the control area, is defined as the distance along the generation boundary from some conveniently chosen reference point. The generation boundary is defined by a set of straight line segments. A typical generation boundary is shown in Figure 4.8.

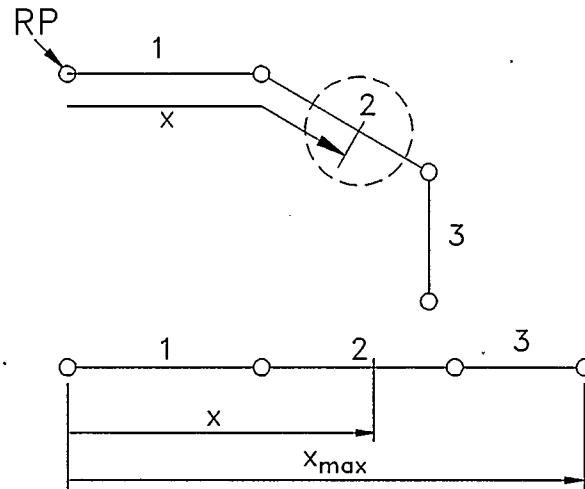


Figure 4.8. Sampling for the position of a rigid body.

The reference point (RP) is chosen at one end of the boundary. The maximum position $x_{\max}$ is then equal to the sum of the lengths of the individual line segments that make up the boundary.

It is assumed that the probability that the generated rigid body occupies any position along the boundary is the same. Hence the random variable x representing position is distributed uniformly over interval $(0, x_{\max})$, and thus the corresponding probability density function is constant. Here,

$$f(x) = \frac{1}{x_{\max}} \quad (4.33)$$

The cumulative distribution function, by Equation (4.4), is therefore

$$F(x) = \frac{x}{x_{\max}}$$

The above result suggests sampling by variable transformation. We use the PRN generator to produce a random number ξ . Then by Equation (4.19) we have

$$\xi = \frac{x}{x_{\max}} \Rightarrow x = \xi \cdot x_{\max} \quad (4.34)$$

Variable x in Equation (4.34) represents the randomly sampled position of the generated rigid body along the generation boundary. This position corresponds to the coordinates of the center of the disk (x_0, y_0) that represents the rigid body.

4.4.3 Sampling for the time interval

We assume that generations of rigid bodies are, in terms of time, Poissonian events or Poissonian walk. Hence, the time increments between consecutive generations are governed by the Poisson distribution.

Suppose an event occurs randomly but on average λ times per unit time. The probability that exactly k events will occur during time t is given by the Poisson distribution, defined by

$$P\{N = k \mid T = t\} = \frac{(\lambda t)^k}{k!} \cdot e^{-\lambda t} \quad (4.35)$$

The probability that no events occur during the time interval t will thus be

$$P\{N = 0 \mid T = t\} = e^{-\lambda t}$$

since $0! = 1$. Now the probability that at least one event will take place during time interval t is obviously one less the probability that no events occur, i.e.

$$P\{N \geq 1 \mid T = t\} = 1 - e^{-\lambda t} \quad (4.36)$$

If we want the Poisson distribution to simulate a random

process such as the generation of rigid bodies, then what we want is to generate randomly "approximate waiting" times for the next event to occur. If T represents a random variable of time increment to the next event, then we want to define its cumulative distribution, i.e. the probability that T is less than t , $(P\{T < t\})$. But T can be less than t only if at least one event occurs in time t , hence using Equation (4.36) we write

$$F_T(t) = 1 - e^{-\lambda t} \quad (4.37)$$

Now if ξ is the generated PRN then by Equation (4.21)

$$\xi = 1 - e^{-\lambda t} \Rightarrow t = -\frac{1}{\lambda} \ln(1 - \xi)$$

and since the distribution of $(1 - \xi)$ is the same as the distribution of ξ we have

$$t = -\frac{1}{\lambda} \ln(\xi) \quad (4.38)$$

Equation (4.38) governs the sampling for the time increment to the next generation of a rigid body. It is evident that the time increment can vary from zero to positive infinity, $0 < t < \infty$. Accepting very large time increments between consecutive generations may not be very practical in terms of the computer simulation. Hence, we wish to sample the time

increments from some finite interval $(t_{\min}, t_{\max})$, and still retain the Poisson distribution.

It can be easily demonstrated that the average time between the consecutive events for a Poisson distribution is $t_{ave} = E(t) = \frac{1}{\lambda}$. The expected value of t , $E(t)$ was defined in Sec.(4.4.1). This is also expected from the definition of parameter λ .

We would like to set up our interval in such a way that the average time remains $\frac{1}{\lambda}$. Thus,

$$E(t) = \frac{1}{\lambda} = \int_{t_{\min}}^{t_{\max}} t \cdot f(t) dt \quad (4.39)$$

and with the use of Equation (4.3) and Equation (4.37) we may write

$$\frac{1}{\lambda} = \int_{t_{\min}}^{t_{\max}} \lambda t \cdot e^{-\lambda t} dt \quad (4.40)$$

Hence,

$$\frac{1}{\lambda} = \frac{1}{\lambda} \cdot \left[- (1 + \lambda t) e^{-\lambda t} \right] \Big|_{t_{\min}}^{t_{\max}}$$

The relation between the endpoints of the time interval, $t_{\min}$, $t_{\max}$ is therefore given by

$$(1 + \lambda t_{\min}) e^{-\lambda t_{\min}} = 1 + (1 + \lambda t_{\max}) e^{-\lambda t_{\max}} \quad (4.41)$$

If we assume that λ , $t_{\max}$ are specified as inputs then the RHS of Equation (4.41) is constant and we have,

$$\lambda t_{\min} = \ln(1 + \lambda t_{\min}) - \ln(C) \quad (4.42)$$

where C is equal to the RHS of Equation (4.41). The above equation can be easily solved numerically by a fixed point iterative procedure to yield $t_{\min}$.

Once the interval $(t_{\min}, t_{\max})$, from which the time increments between the consecutive events are sampled, has been established, the cumulative distribution function can be adjusted so that $F_T(t_{\min}) = 0$, $F_T(t_{\max}) = 1$. Hence,

$$F_T(t) = A[1 - \exp(-\lambda(t - t_{\min}))]$$

where

$$A = \frac{1}{1 - \exp(-\lambda(t_{\max} - t_{\min}))}$$

Finally, the relation governing sampling is given by

$$t = t_{\min} - \frac{1}{\lambda} \ln\left(1 - \frac{\xi}{A}\right) \quad (4.43)$$

The above equation is the modified form of Equation (4.38) and random variable t sampled according to it, will have a Poisson distribution and will fall within $(t_{\min}, t_{\max})$. The appropriate routine was programmed on the computer.

4.4.4 Positioning of the generated rigid bodies

The generated rigid bodies have to be positioned on the generation boundary in such a way that they do not overlap with any other rigid bodies already in the system, or with any of the obstruction line segments. The methodology of testing for the overlap and the subsequent adjustment of the position of the generated rigid body will be presented in this subsection.

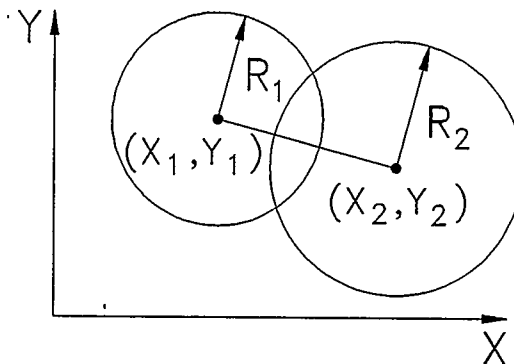


Figure 4.9. Overlap between two disks.

The overlap between two disks can be easily tested for. The radii of the disks are known and so are the coordinates of their centers. Fig.(4.9) shows a typical situation. The overlap exists if the following condition is satisfied:

$$\sqrt{(X_2 - X_1)^2 + (Y_2 - Y_1)^2} \leq R_1 + R_2 \quad (4.44)$$

In the limiting case when condition (4.44) becomes an equality, the overlap consists of just one point.

Testing for the overlap of a disk with a line segment $(\overline{a,b})$ is shown in Fig.(4.10).

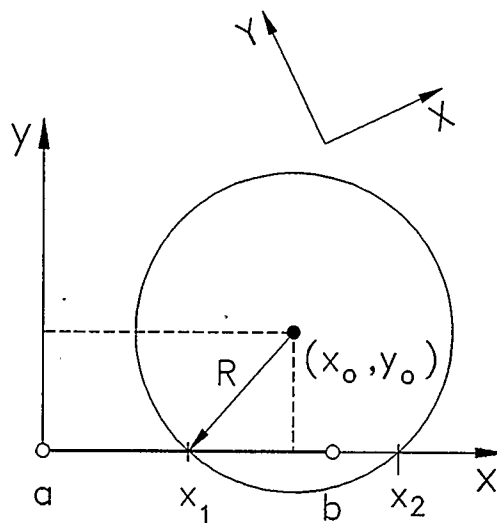


Figure 4.10. Overlap of a disk with a line segment.

It is convenient to perform the testing in the local system of coordinates, with the origin located at one of the endpoints and with the x -axis coinciding with the line segment. The coordinates of the center of the disk are transformed from the global to the local coordinate system. Since our program already evaluates the unit normal vector $\hat{n}$ for every line segment of the solid boundary, those will be used to define the transformation. We will use upper case letters to denote

the global coordinate system and lower case letters for the local system of coordinates. Let n_x, n_y be the components of the unit normal vector of the straight line segment $(\overline{a, b})$ in the global system of coordinates. If the coordinates of the endpoints of the segment are X_a, Y_a, X_b, Y_b , then the above transformation is given by

$$\left. \begin{aligned} x &= n_y (X - X_a) - n_x (Y - Y_a) \\ y &= n_x (X - X_a) + n_y (Y - Y_a) \end{aligned} \right\} \quad (4.45)$$

Clearly, the transformed coordinates of the endpoints of the line segment are: $x_a = 0, y_a = 0, x_b = l_{ab}, y_b = 0$, where

$$l_{ab} = |(\overline{a, b})| = \sqrt{(X_b - X_a)^2 + (Y_b - Y_a)^2}$$

If x_0, y_0 denote the transformed coordinates of the center of the disk then, clearly, an overlap with a line segment can occur only if

$$y_0 \leq R \quad (4.46)$$

If condition (4.46) is satisfied then the circle of Figure 4.10 intersects the x -axis. There are two points of intersection which we will denote as x_1, x_2 . If equality in condition (4.46) holds then the two points will coincide as the circle will be tangent to the axis. The equation of the circle in Figure 4.10 is given by

$$(x - x_0)^2 + (y - y_0)^2 = R^2$$

The intercept points can be evaluated by setting $y = 0$.
Hence,

$$x_{1,2} = x_0 \pm \sqrt{R^2 - y_0^2} \quad (4.47)$$

Finally, the overlap between the disk and the solid boundary line segment occurs if

$$\{(a, b) \cap (x_1, x_2)\} \neq \emptyset$$

which is equivalent to at least one of the following two conditions being satisfied:

$$\left. \begin{array}{l} 1) \quad x_1 \in (a, b) \\ 2) \quad x_2 \in (a, b) \end{array} \right\} \quad (4.48)$$

If an overlap is detected for a generated disk (X_0, Y_0, R) then the position of this disk has to be adjusted so that no overlap occurs. Since the position is represented by a uniformly distributed random variable, the generality of the model will not be jeopardised if this variable is adjusted. Let us consider the example shown in Fig.(4.11).

In the figure the shaded lines represent the solid boundary and the straight line connecting points A and B is the

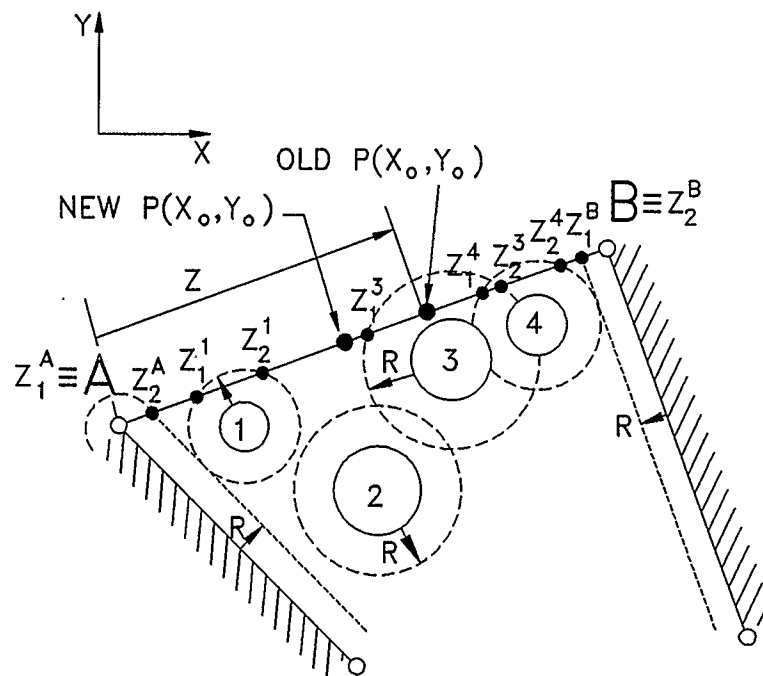


Figure 4.11. Definition of the overlap zone and overlap intervals.

generation boundary. There are already four disks present in the system, numbered 1 to 4. The broken-line involute around the disks and the solid boundary line segments encloses what is termed as an "overlap zone", i.e., an area in the system where a generated disk of size R cannot be positioned without a resulting overlap. It is clear that the involute is at a distance R from the surfaces of the respective bodies. There are six individual overlap zones. These result from four disks and two solid boundary line segments. The overlap intervals are defined as the areas on the generation boundary which lie inside the overlap zones. In other words, the overlap intervals are the areas of the generation boundary

where the generated disk with radius R can not be positioned. In our example there are five overlap intervals, three from disks 1, 3, and 4, and two from the solid boundary at points A and B. If z represents the distance along the generation boundary from its reference point (point A here), then the overlap intervals can be defined as the subsets of $\overline{AB}$. The subscripts 1 and 2 are used to denote the endpoints of the respective overlap intervals, while the superscripts identify the bodies creating them. In the example shown those are

$$(z_1^A, z_2^A), (z_1^1, z_2^1), (z_1^3, z_2^3), (z_1^4, z_2^4), (z_1^B, z_2^B)$$

Since the overlap intervals from disks 3 and disk 4 overlie, they can be combined into one, i.e., $(z_1^3, z_2^3) \cup (z_1^4, z_2^4) = (z_1^3, z_2^4)$. Hence, in the example shown there are four resulting discrete overlap intervals. In general there could, of course, be more of such discrete intervals.

If a would-be overlap is detected for the generated disk then all the overlap intervals from the bodies already in the system are established. Consequently, the generated position z (see Section 4.4.2) is adjusted in such a way that it falls outside any of the identified overlap intervals. The adjustment is also made so that the adjusted position is in the closest proximity to the original one. Hence, the disk generated at the new position will not overlap, either with

any of the other disks already present in the system, or with the line segments of the solid boundary. If a situation arises, where the overlap extends over the entire generation boundary, then the generation of a rigid body is skipped for this instant in time. The time increment to the new generation is however retained.

The methodology behind the procedure identifying the overlap intervals will not be presented here. It largely involves geometrical manipulations of circles and straight line segments. The appropriate computer routines were programmed and tested. These are described in Appendix D.

4.5 SURFACE DENSITY CONCEPT

Let us suppose that using the methodology described in Section 4.4 we generated say, N disks. Let us also assume that after the generation the disks moved away from the generation boundary at a constant rate. With our choice of parameters σ, μ, λ , what is the surface density of the disks in the system? We define surface density as the ratio of the total area of the disks to the control area that these disks occupy. We write

$$\rho = \frac{A_D}{A_{TOT}} = \frac{1}{A_{TOT}} \left[\sum_{i=1}^n \pi x_i^2 \right] \quad (4.49)$$

where x_i denotes the size (radius) of a disk.

Let us now consider a situation shown in Figure 4.12. We assume that the generation boundary ($\overline{AB}$) has length L and the velocity with which the generated disks are moving away is v . If the generation is being carried out over some time interval T , then assuming that $T > \frac{1}{\lambda}$, the approximate number of disks in the system is given by:

$$N_T = \lambda T \quad (4.50)$$

We assume that the areas of the disks y_i are distributed

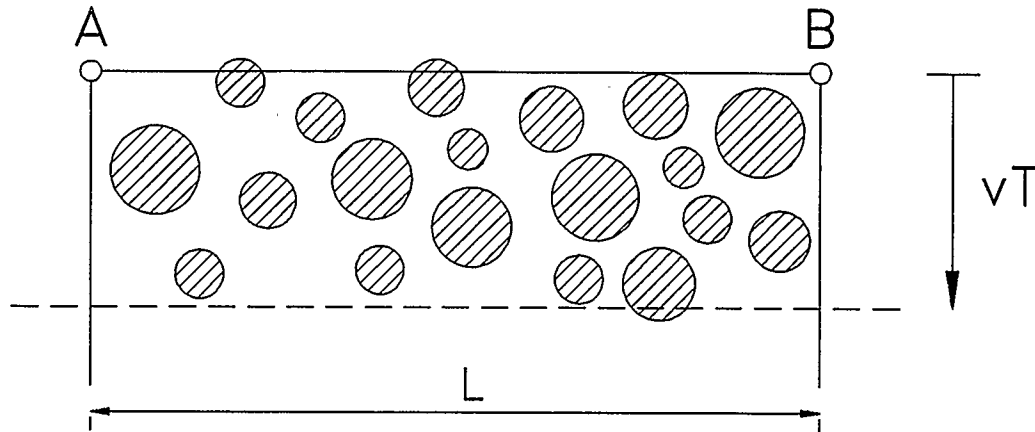


Figure 4.12. Diagram for evaluating the surface density.

according to a probability density function $f_Y(y)$. If we consider a small disk area increment δy then the number of disks with areas which fall within this interval, using Equation (4.14), is

$$\delta N_{\delta y} = N_T \cdot P\{y \in \delta y\} = N_T \cdot f_Y(y) \delta y \quad (4.51)$$

The total area occupied by these disks is clearly the number of disks multiplied by their size, . In the limiting case this relation can be written as

Using the result of Equation (4.50) we can write an expression for the total area occupied by the disks .

$$A_D = \lambda T \left[\int_{y_{\min}}^{y_{\max}} y f_Y(y) dy \right] \quad (4.52)$$

The term in square brackets in the above equation represents the expected or mean value $E(y)$, and since the total control area which encloses the disks is $A_{TOT} = LvT$, we can write an expression for the surface density as:

$$\rho = \frac{\lambda E(y)}{vL} \quad (4.53)$$

The expected value $E(y)$ in the above equation can be determined since $y = y(x)$ and the distribution for the random variable x , $f_X(x)$ is known. Namely, since $y(x) = \pi x^2$ is, for positive values, a non-decreasing function of x , then the corresponding cumulative distributions are equal, i.e., $F_Y(y) = F_X(x)$, (see Section 4.3). And hence we may write,

$$f_Y(y) dy = f_X(x) dx$$

Substituting the above result into Equation (4.52) we obtain:

$$E(y) = \pi \int_{x_{\min}}^{x_{\max}} x^2 f_X(x) dx \quad (4.54)$$

where is given by Equation (4.32).

If our input comprises of parameters μ , σ , ρ , then the integral of Equation (4.54) can be evaluated numerically and the Poissonian constant λ can be determined so that the desired surface density of the disks is achieved. Namely,

$$\lambda = \frac{\rho v L}{E(y)} \quad (4.55)$$

As a final note it should be mentioned that if the maximum time increment to the next generation $t_{\max} < \frac{1}{\lambda}$, then the program will automatically set $t_{\max} = 4 \cdot \frac{1}{\lambda}$ and the minimum time increment will be evaluated according to Equation (4.42).

4.6 OTHER RELATED TOPICS

4.6.1 Other shapes for rigid bodies

A variety of shapes can be composed from disks. At the present, we shall limit ourselves to the consideration of equilateral triangles and squares. Figure 4.13 shows the two shapes under consideration.

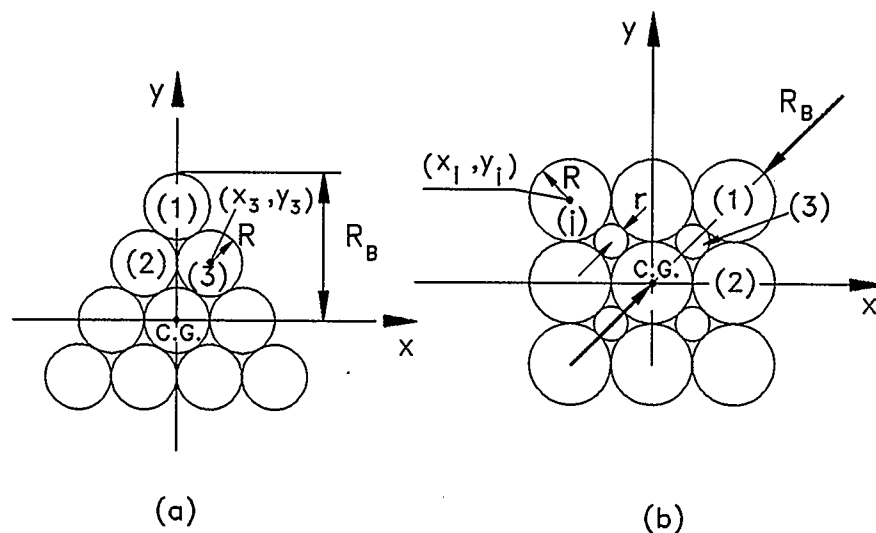


Figure 4.13. Different shapes of rigid bodies, triangle (a), and square (b).

A triangle is composed of ten equal disks, while a square is composed out of nine large and four small disks. It is evident from the geometry of the shapes that if we maintain the connectivity among the corresponding disks, as shown in Figure 4.13, then the entire configuration will remain

unchanged, and hence such a body will be, indeed, rigid. In terms of our computer model this means that the specified connections among the disks will not be cancelled at any time in the program. Consequently, the shapes of such bodies will not be altered throughout the simulation.

The generation of complex bodies is performed in very similar way to that for the simple bodies, (Section 4.4). We again have three random variables: size, position, and the time increment. These random variables are sampled according to the procedures presented in Section 4.4.

The random variable of position corresponds to a point on the generation boundary denoted as (X_0, Y_0) . The generated complex body is positioned on the generation boundary in such a way that its center of gravity coincides with this point. From the geometry of the bodies shown in Figure 4.13 it becomes evident that the respective centers of gravity for triangular and square bodies are at the centers of the corresponding central disks (C.G.). At the time of generation the central disk will, thus, occupy the position defined by coordinates (X_0, Y_0) . We now need to establish the coordinates of the centers of all other disks comprising the body (X_i, Y_i) . This can be accomplished by first defining the coordinates relative to the body's center of gravity in a local system of coordinates. The origin of such a system of

coordinates is located at the center of gravity of the body, as shown in Figure 4.13. We will denote local coordinates as (x_i, y_i) . We now define the expression for the transformation from a local into the global coordinate system. We write

$$\left. \begin{aligned} X_i &= x_i + X_0 \\ Y_i &= y_i + Y_0 \end{aligned} \right\} \quad (4.56)$$

The local coordinates depend only on the radii of the disks comprising the body and thus, can be readily computed. For example, the local coordinates of the center of disk 3 of the triangular body are $x_3 = R$, $y_3 = R\sqrt{3}$. The global coordinates by Equation (4.56) are thus $X_3 = X_0 + R$, $Y_3 = Y_0 + R\sqrt{3}$.

The random variable representing the size of a complex body corresponds to a distance from the body's center of gravity to the most outer point on the body. This variable is denoted as R_B in Figure 4.13 and is generated by the program. From the figure it can be seen that for a triangular body $R_B = R(1 + 2\sqrt{3})$. Similarly, for a square body we have $R_B = 3R + 2r$. From the geometry of the square body we can establish the relation between the radii of small and large disks as: $r = R(\sqrt{2} - 1)$. Now the radii of the disks that comprise both triangular and square bodies can be computed in terms of R_B . For a triangular body

$$R = \frac{R_B}{2\sqrt{3} + 1} \quad (4.57a)$$

and for a square body,

$$\left. \begin{aligned} R &= \frac{R_B}{1 + 2\sqrt{2}} \\ r &= R_B \cdot \frac{\sqrt{2} - 1}{1 + 2\sqrt{2}} \end{aligned} \right\} \quad (4.57b)$$

This concludes the definition of a complex body. Upon the generation of R_B the radii of the disk can be determined from Equation (4.57). Consequently, the body can be positioned on the generation boundary and the coordinates of the centers of all component disks established by Equation (4.56).

4.6.2 Generation of random shapes

Two-dimensional random shapes can be generated as polygons whose vertices are randomly generated points on the plane. Perhaps the easiest way of achieving this is to generate a set of random points given by cylindrical coordinates (r_i, θ_i) . These coordinates correspond to pairs of uniformly distributed random numbers (ξ_i, η_i) which are obtained from the square PRN generator (Section 4.2). The governing relation is given by:

$$\left. \begin{aligned} r_i &= r_{\min} + \xi_i (r_{\max} - r_{\min}) \\ \theta_i &= 2\pi \eta_i \end{aligned} \right\} \begin{aligned} r_i &\in (r_{\min}, r_{\max}) \\ \theta_i &\in (0, 2\pi) \end{aligned} \quad (4.58)$$

where $r_{\min}, r_{\max}$ are the chosen limits for the radial coordinate. An example in Figure 4.14 illustrates a situation where ten points were generated.

The generated points are sorted according to the tangential coordinate θ , in ascending order. The corresponding points are then joined in this order to create a polygon.

For the example shown in Figure 4.14 the random shape is given thus by a polygon established from the points in the

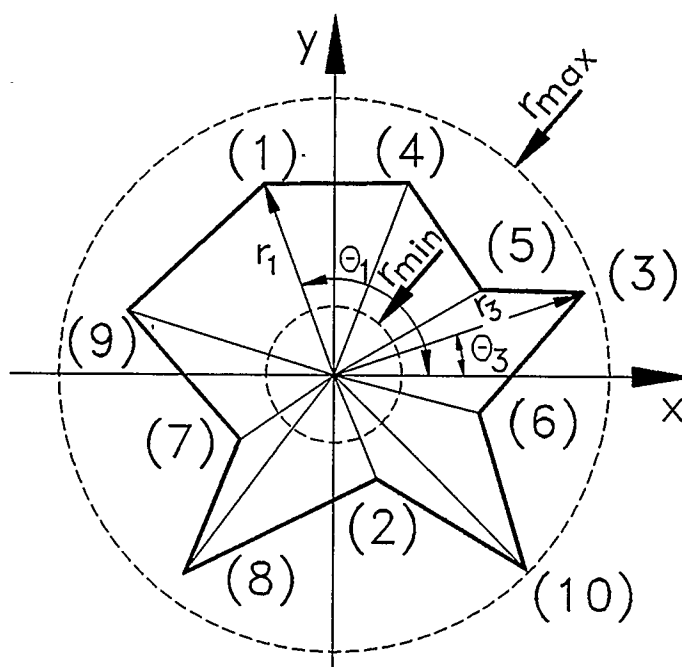


Figure 4.14. A polygon type two dimensional random shape.

following order 3-5-4-1-9-7-8-2-10-6-3.

Another way of obtaining a random shape is by generating a number of disks having random radii which are then connected randomly in such a manner that the resulting geometrical figure is a rigid body. Such rigid bodies are more suitable for our model, although the polygon type shapes can also be handled. Figure 4.15 shows an example of random shape created with a set of twelve disks.

The method of creating randomly shaped rigid bodies

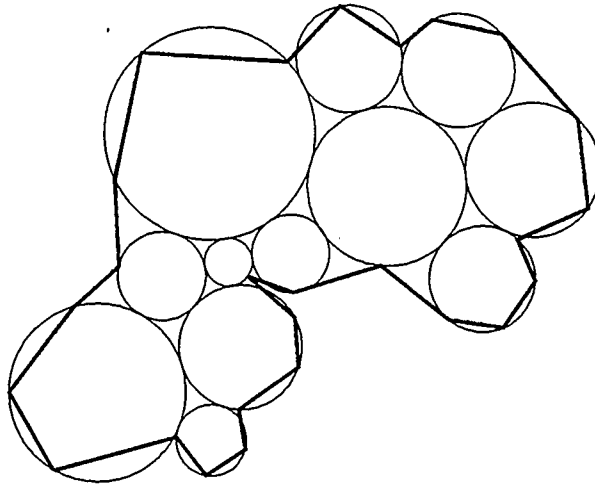


Figure 4.15. A random shape rigid body created from disks of random size.

composed of disks, which was developed for the purpose of our model, consists of generating the individual disks one at a time and randomly adding them to the existing body in such a way that the rigidity of the geometrical figure is maintained. Such addition is achieved by placing a given disk in contact with a randomly chosen pair of neighbouring disks which are already a part of the body.

Figure 4.16 shows the process of creating a random shape rigid body from randomly generated disks. We start with three disks of random size arranged in a triangle as shown in Figure 4.16a. There are three pairs of neighbouring disks to which the next disk can be added, namely 1-2, 1-3, 2-3. We now randomly generate the radius of the next disk (4) and randomly

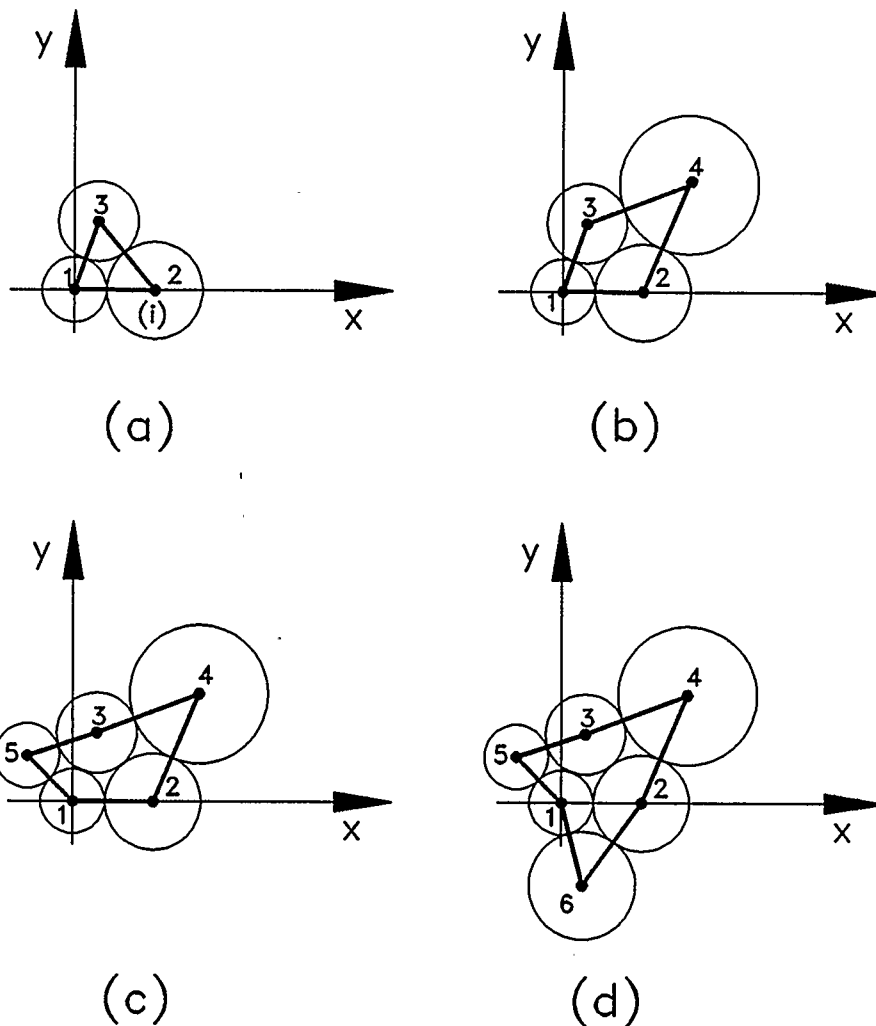


Figure 4.16. An example of creating a random shape rigid body.

choose one of the three pairs of neighbouring disks, say 2-3. We add the new disk (4) to the body by positioning it in contact with disks (2) and (3), Figure 4.16b. Now there are four disks composing the body. There are also four pairs of neighbouring disks to which the next disk can be added. Those

are indicated in the figure as line segments connecting the centers of the respective disks.

Similarly we add disk (5), Figure 4.16c, and disk (6), Figure 4.16d. The procedure is repeated until the body is composed of twelve disks. Each time a disk is added to the body its local coordinates are evaluated. The local system of coordinates is chosen in such a way that its origin is at the center of disk (1) and its x-axis runs along the line connecting the centers of disks (1) and (2), as shown in Figure 4.16.

For the purpose of our model the radii of the disks, R_i , comprising the rigid body are randomly picked from (0.5,1) interval. For this we use the PRN generator of Section 4.2. If ξ represents a random number generated from (0,1) interval then the corresponding radius of a disk is given by

$$R = 0.5 + \frac{\xi}{2} \quad (4.58)$$

The pair of neighbouring disks to which the new disk is attached is randomly chosen from a set of all such possible pairs. Each pair of disks, to which the addition of a new disk is possible, is numbered from 1 to N , where N represents the total number of such pairs. Consequently, the selection of a pair of disks is reduced to randomly choosing an integer

between 1 and N . Again, we use the PRN generator of Section 4.2 to generate a random number $\eta \in (0, 1)$, and we select a pair of disks according to

$$i = 1 + \text{INT}(N \cdot \eta) \quad (4.59)$$

All the pairs of disks are stored in a two-dimensional incidence matrix $\text{IRS}(2, N)$. This matrix has N columns which correspond to the numbers of disk pairs, and each column has two elements which store the numbers of the disks comprising the pair. For example, elements $\text{IRS}(1, j)$ and $\text{IRS}(2, j)$ store the numbers of the two disks belonging to disk pair j . For the situation shown in Figure 4.16c the incidence matrix would be

$$\begin{bmatrix} 1 & 2 & 4 & 3 & 5 \\ 2 & 4 & 3 & 5 & 1 \end{bmatrix}$$

From the above matrix, for example, disk pair 3 which corresponds to the third column, is thus created by disks 3 and 4.

An addition of a new disk to the given rigid body consists of generating a pair of random numbers (ξ, η) . The size of the disk and its relative position in the body (i.e., with which two disks it is to be in contact) are determined from Equations (4.58) and (4.59). The coordinates of the center of the new disk can now be computed. Let us assume

that the new disk of radius R is to be placed in contact with two neighbouring (in contact) disks, i, j , with respective radii and coordinates of their centers being (R_i, x_i, y_i) , (R_j, x_j, y_j) , as shown in Figure 4.17.

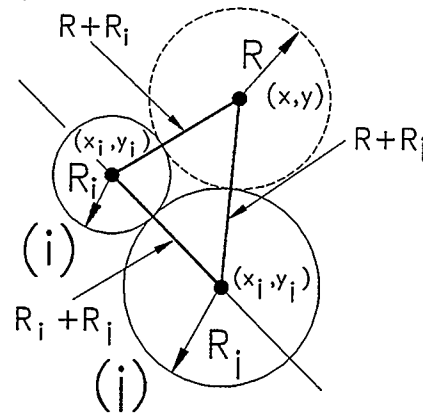


Figure 4.17. Calculation of coordinates of the center of a new disk being added to the rigid body.

If the new disk is to be in contact with the two disks then the equations governing the coordinates of the new disk (x, y) are given by

$$\left. \begin{aligned} (x - x_i)^2 + (y - y_i)^2 &= (R + R_i)^2 \\ (x - x_j)^2 + (y - y_j)^2 &= (R + R_j)^2 \end{aligned} \right\} \quad (4.60)$$

We define two constants,

$$a_1 = (R + R_i)^2 - x_i^2 - y_i^2, \quad a_2 = (R + R_j)^2 - x_j^2 - y_j^2$$

Now, the coordinates x, y are evaluated as follows:

if $y_j \neq y_i$ then

$$\left. \begin{aligned} x &= -\frac{B}{2A} \pm \frac{\sqrt{B^2 - 4AC}}{2A} \\ y &= \frac{a_1 - a_2}{2(y_j - y_i)} - x \cdot \frac{x_j - x_i}{y_j - y_i} \end{aligned} \right\} \quad (4.61)$$

where:

$$\begin{aligned} A &= 1 + \left(\frac{x_j - x_i}{y_j - y_i} \right)^2 \\ B &= -2x_i - \frac{(a_1 - a_2) \cdot (x_j - x_i)}{(y_j - y_i)^2} + 2y_i \cdot \frac{x_j - x_i}{y_j - y_i} \\ C &= \left(\frac{a_1 - a_2}{2(y_j - y_i)} \right)^2 - y_i \cdot \frac{a_1 - a_2}{y_j - y_i} - a_1 \end{aligned}$$

and if $y_j = y_i$ then,

$$\left. \begin{aligned} x &= \frac{1}{2} \frac{a_1 - a_2}{x_j - x_i} \\ y &= y_i \pm \sqrt{y_i^2 + a_1 + 2xx_i - x^2} \end{aligned} \right\} \quad (4.62)$$

The plus-minus signs in Equation (4.61) and Equation (4.62) correspond to the two possibilities of positioning a disk in contact with two other disks. In terms of generating the rigid body, these possibilities represent the inside and the outside of the contour polygon (shown as solid lines in Figure

4.16. Consequently, the disk is positioned to the outside of the polygon.

Once the new disk is attached to the pair of neighbouring disks, that pair cannot accept any more disks, and hence is deleted from a set of possible pairs. However, the addition of the disk creates two new pairs which are added to the set. These involve the new disk and each of the two disks that formed the original pair. If we use the example shown in Figure 4.16, then for part (c) the incidence matrix is

$$\begin{bmatrix} 1 & 2 & 4 & 3 & 5 \\ 2 & 4 & 3 & 5 & 1 \end{bmatrix}$$

If another disk (6) is added to the rigid body as shown in part (d) of the figure, then the disk pair 1-2 is deleted and two new pairs, 1-6 and 2-6, are added in its place. The incidence matrix would thus become

$$\begin{bmatrix} 1 & 6 & 2 & 4 & 3 & 5 \\ 6 & 2 & 4 & 3 & 5 & 1 \end{bmatrix}$$

This concludes the description of the methodology of creating the randomly shaped rigid bodies out of disks. The shape of such a body will remain unchanged as long as the specified connections among the disks are maintained.

Randomly shaped rigid bodies are treated by the program

in the same way that are the square or the triangular bodies discussed in Section 4.6.1. After the body is created it can be entered into the system (i.e., positioned on the generation boundary) in very much the same way. We again have two generated random variables: the size and the position on the generation boundary. These two variables are always maintained by the model regardless of whether the body is complex or simple.

As indicated before the random variable of position which is generated by the program corresponds to the point on the generation boundary (X_0, Y_0) at which the center of gravity of the generated body (simple or complex) is positioned. For the randomly shaped rigid body the location of the center of gravity is not known beforehand and has to be computed. Since the coordinates of the centers of all disks (x_i, y_i) and their respective radii (R_i) are known in the local system of coordinates (see Figure 4.16), the position of the center of gravity of the body (x_c, y_c) can be determined from the following relation:

$$x_c = \frac{\sum_{i=1}^N x_i \pi R_i^2}{\sum_{i=1}^N \pi R_i^2}, \quad y_c = \frac{\sum_{i=1}^N y_i \pi R_i^2}{\sum_{i=1}^N \pi R_i^2} \quad (4.63)$$

where N is the total number of disks comprising the rigid

body.

The coordinates of the centers of individual disks with respect to the body's center of gravity (x'_i, y'_i) can now be computed from a simple transformation,

$$\left. \begin{aligned} x'_i &= x_i - x_c \\ y'_i &= y_i - y_c \end{aligned} \right\} \quad (4.64)$$

The generated random variable of size (R_B) , which is assigned to the rigid body, corresponds to the distance from the body's center of gravity to the most outer point on its surface. Since the radii of the disks comprising such a body are randomly picked values between 0.5 and 1, it is very unlikely that for the created random shape rigid body the maximum distance will coincide with the previously generated size. Consequently, the given rigid body has to be rescaled, so that it fits the proper size. If we denote the scaling factor as k_s then we can write

$$R_B = k_s \cdot \max \left\{ R_i + \sqrt{(x'_i)^2 + (y'_i)^2}, \quad (i=1, N) \right\}$$

The maximum value of the expression in braces can be evaluated for any given body and hence, the scaling factor can be determined,

$$k_s = \frac{R_B}{\max \left\{ R_i + \sqrt{(x'_i)^2 + (y'_i)^2} \right\}} \quad (4.65)$$

For the rescaled body the radii of the disks (R_i^*) and the coordinates of their centers with respect to body's center of gravity (x_i^*, y_i^*) can be evaluated,

$$R_i^* = k_s R_i, \quad x_i^* = k_s x_i, \quad y_i^* = k_s y_i \quad (4.66)$$

The given random shape rigid body represented by a set of disks is now defined in a proper context and can now be positioned on the generation boundary. The radii of the disks are given by (R_i^*) and the global coordinates of the respective centers (X_i, Y_i) are determined by the following coordinate transformation:

$$\left. \begin{aligned} X_i &= x_i^* + X_0 \\ Y_i &= y_i^* + Y_0 \end{aligned} \right\} \quad (4.67)$$

This concludes the process of generation of the random shape rigid body composed of a set of random disks.

4.7 CONCLUSIONS

The method of rigid body generation presented in this chapter was successfully implemented in the model.

The generated rigid bodies can have a variety of shapes, ranging from circular, square, and triangular to random ones. The overall sizes of the rigid bodies can be randomly sampled according to any chosen distribution function, both continuous and discrete. All considered, the developed method of rigid body generation provides a user with a very versatile tool in the modelling of stochastic type processes, involving the dynamics of solid-fluid systems.

The PRN generator chosen in Section 4.2 proved to be versatile in generating the sequences of single numbers and the pairs of numbers with good statistical properties.

The method also allows for the random generation of rigid bodies in such a way that overall, a certain specified surface density of rigid bodies is retained throughout the simulation process.

CHAPTER 5

FLUID FLOW VELOCITY FIELD

5.1 INTRODUCTION

Since the simulation model is also directed at analyzing the motion of multi-body systems in a fluid medium, it is necessary to consider the interactions between the solids and the surrounding fluid. These interactions occur at the solid-fluid interfaces and are governed by the relative velocities of one with respect to the other. The existence of such differences in velocities gives rise to the interaction forces in the form of drag and lift, which in turn affect the motion of the solid bodies.

If multi-body systems are to be analyzed in a stream of moving fluid, then the velocity field of the fluid flow is a necessary input for the model. This chapter describes a set of computer routines which were developed for the computation of the flow velocity fields. The free stream velocity and the geometry of the solid boundaries are the initial inputs. The flow is assumed to be two-dimensional. Both a flow around an obstruction as well as the channel flow are considered.

The general equations governing the motion of the fluid, as we shall see in the next section, are very complicated and solving them presents considerable mathematical difficulty. These equations, however, can be greatly simplified with the assumptions of incompressibility and irrotationality of the flow. The two assumptions bring us to what is known as the potential flow theory and the corresponding equations of motion are readily solved.

Many types of fluid flow can be considered as being potential, such as for example, a uniform flow approaching a structure, a channel flow around a bend, or any type of converging flow. There are flows, however, for which the potential theory does not apply. These include boundary layers, wakes, jets, and the areas of significant boundary layer separation.

Frequently interactions between a structure and solid bodies flowing with the fluid and the loads exerted on the structure due to those interactions may be of interest. The solid-structure interactions occur mainly on the part of the structure's boundary which is facing the flow. In such instances the flow can be considered irrotational and, hence, potential flow theory is applicable.

5.2 EQUATIONS OF MOTION

The equations of motion for a fluid particle are formulated in accordance with the basic laws of mechanics - conservation of mass and energy, and Newton's laws of motion. These equations involve physical and kinematic properties of the fluid, such as velocity, pressure, density, and temperature, which are assumed to vary continuously throughout the fluid. In order to be able to define these properties at any point in the fluid one must make an assumption that the fluid is a continuous medium and that continuum mechanics is, in general, applicable. Consequently, it is assumed that at each point in the fluid there is a fluid particle and that each volume of fluid, no matter how small, contains an infinite number of such particles.

The first principal equation governing the fluid flow is the conservation of mass, also known as the continuity equation. This condition can be written as (see Appendix E):

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{u}) = 0 \quad (5.1)$$

where ρ is fluid's density and $\vec{u}$ the velocity vector of a fluid particle. For an incompressible flow (i.e. $\rho = \text{const}$) the continuity equation reduces to a very simple form,

$$\nabla \cdot \vec{u} = 0 \quad (5.2)$$

An interesting observation can be made here. Time does not enter the continuity equation for an incompressible fluid flow explicitly, even if the flow itself is unsteady.

The second fundamental equation of motion is the momentum equation which for a Newtonian fluid may be written as (Appendix E):

$$\rho \frac{d\vec{u}}{dt} = -\nabla p + (\lambda + \mu) \nabla (\nabla \cdot \vec{u}) + \mu \nabla^2 \vec{u} + \rho \vec{b} \quad (5.3)$$

Equation (5.3) is known as the generalized Navier-Stokes equation for a Newtonian fluid. In the equation μ is the coefficient of viscosity and λ is the second viscosity coefficient. It is common engineering practice to relate the two independent viscosity coefficients according to Stokes condition ($\lambda = -\frac{2}{3}\mu$) since the second viscosity is difficult to measure and is not known for many fluids. In the equation the term d/dt is referred to as the material time derivative and is given by the following relation:

$$\frac{d}{dt} = \frac{\partial}{\partial t} + \vec{u} \cdot \nabla \quad (5.4)$$

If fluid incompressibility is assumed, then by Equation (5.2) the term involving λ vanishes from the momentum equation (5.3). Hence, the Navier-Stokes equations for an

incompressible fluid are given by:

$$\rho \frac{d\vec{u}}{dt} = -\nabla p + \mu \nabla^2 \vec{u} + \rho \vec{b} \quad (5.5)$$

Equations (5.2) and (5.5) constitute the principal equations of motion for an incompressible fluid. A detailed derivation of these equations can, readily, be found in many textbooks on fluid dynamics or continuum mechanics (see for example [4], [25], [34]).

It can be noticed that Equation (5.5) is a non-linear partial differential equation in $\vec{u}$. The non-linearity appears in the material time derivative of the velocity vector (inertial acceleration term: $\rho d\vec{u}/dt$), and interestingly, it does not vanish even when ideal, frictionless, fluid is considered. It is due to this non-linearity that there are no known general solutions to the Navier-Stokes equations. It was not until recently, with the considerable developments in numerical methods and digital computers that the solutions to the generalized Navier-Stokes equations were obtained. Most commonly, the solutions are based on finite element or finite difference techniques [4], [10]. These methods in themselves present a considerable mathematical difficulty and are well beyond the scope of the present thesis.

As a final note we will introduce a quantity called a

vorticity vector $\vec{\omega}$ and defined as the *curl* of the velocity vector field:

$$\vec{\omega} = \nabla \times \vec{u} \quad (5.6)$$

As we shall see in the next section an irrotational flow, i.e. with zero vorticity, represents a special type of flow for which the corresponding equations of motion can be solved much more readily than the full Navier-Stokes equations. Many types of real flow can be considered irrotational and hence, the assumption of irrotationality (i.e. $\vec{\omega} = 0$) can prove to be very useful in flow computation.

5.3 POTENTIAL FLOW THEORY

Under certain conditions the motion of a fluid can be described mathematically by potential functions. A well known theorem of vector calculus states that a curl of a vector field equal to zero is a necessary and sufficient condition for the existence of a scalar potential function such that the vector field can be derived as the gradient of this function. In fluid mechanics this implies that the absence of vorticity in the flow is a necessary and sufficient condition for the existence of a scalar velocity potential.

Hence, if $\vec{\omega} = \nabla \times \vec{u} = \vec{0}$, then there exists a scalar velocity potential ϕ such that

$$\vec{u} = \nabla \phi \quad (5.7)$$

Conversely, if the flow velocity vector field can be expressed in terms of a gradient of a scalar potential function ϕ , as in Equation (5.7), then the flow is irrotational since the vorticity $\vec{\omega} = \nabla \times (\nabla \phi) = \vec{0}$ for any such function.

For incompressible flow the continuity equation, (5.2) can be expressed in terms of a velocity potential. Namely,

$$\nabla \cdot \nabla \phi = \nabla^2 \phi = 0 \quad (5.8)$$

Hence, the velocity potential satisfies the Laplace equation.

The momentum equation for an incompressible and an irrotational flow can also be expressed in terms of a velocity potential. Namely (see Appendix F),

$$\nabla \left(\frac{\partial \phi}{\partial t} + \frac{1}{2} u^2 + \frac{p}{\rho} \right) = \vec{b} \quad (5.9)$$

It is apparent from the above result that a potential flow can occur only in the presence of conservative (potential) body forces. If Π represents a known body force potential then by definition: $\vec{b} = \nabla \Pi$, and Equation (5.9) becomes

$$\nabla \left(\frac{\partial \phi}{\partial t} + \frac{1}{2} u^2 + \frac{p}{\rho} - \Pi \right) = 0 \quad (5.10)$$

This implies that the quantity in parentheses must either be constant or a function of time alone. Hence,

$$\frac{\partial \phi}{\partial t} + \frac{1}{2} u^2 + \frac{p}{\rho} - \Pi = f(t) \quad (5.11)$$

This is known as the generalized Bernoulli equation for an incompressible and an irrotational flow. In a specific case when the body force comprises of only gravity, then in

cartesian coordinates, using the summation convention for repeated indices, the force potential is equal to, $\Pi = -gx_j n_j$, where n_j is a directional cosine of the angle between the j axis of the chosen spatial coordinate system and the vertical axis of the global reference coordinate system.

As was demonstrated, the introduction of the velocity potential replaced the three unknown velocity components with a single scalar function ϕ , thus greatly simplifying the equations of motion. Consequently, with the assumptions that the flow is incompressible and irrotational, the four general equations of motion (one continuity, Equation (5.1), and three momentum, Equation (5.3)) in four unknowns (three velocity components and pressure) were replaced by two equations Equation (5.8) and Equation (5.11), in two unknowns, ϕ, p . A solution to these two equations can be obtained without significant mathematical difficulties. One, however, is compelled to address the question of the validity of the assumptions used and hence, the applicability of the potential flow equations.

It is generally accepted, [14], [16], [34], that the assumption of incompressibility is valid for all flows where the local Mach number does not exceed one half. The assumption that the flow is irrotational is justified for high Reynolds Number ($Re = \rho u l / \mu$) flows outside the boundary layer

and the regions of considerable separation (for example: wakes). The significance of the Reynolds number lies in the fact that it is of the same order of magnitude as the ratio of the inertia forces term ($\rho d\vec{u}/dt$) to the viscous forces term ($\mu \nabla^2 \vec{u}$) in Equation (5.5). High Reynolds numbers would, thus, indicate that the viscous forces are small with respect to the inertia forces and can be neglected. This is equivalent to the assumption that the flow is irrotational (Appendix F, Equation (F.5)). It therefore becomes apparent that the equations of motion derived from the potential flow theory constitute an adequate description of the flows considered in the present model. Furthermore, it is assumed that the flow is steady. Consequently, the time-variant terms in Equation (5.11) will vanish. Also, since our present model is two-dimensional, with events and interactions occurring in the horizontal x-y plane, the term involving the body forces will also disappear. Hence, the final form of the equations of motion used in the present model is

$$\nabla^2 \phi = 0 \quad (5.12)$$

$$\frac{1}{2} u^2 + \frac{p}{\rho} = \text{const} \quad (5.13)$$

Since the viscous effects were neglected the boundary condition at the solid surface is that the local velocity be parallel to the surface. In other words, that the velocity

component normal to the surface be zero (impermeability condition).

$$\frac{\partial \phi}{\partial n} = 0 \quad (5.14)$$

where n indicates a direction normal to the surface. The above condition is equivalent to: $\vec{u} \cdot \hat{n} = 0$. Hence, using Equation (5.12) the boundary condition may be written as:

$$\nabla \phi \cdot \hat{n} = 0 \quad (5.15)$$

The Laplace equation (5.12) together with the boundary condition (5.14) comprise what is known as the Neumann problem.

The exact analytic solutions to the Laplace equation are possible only for some specific axisymmetric and three-dimensional cases. These solutions employ the technique of separation of variables, wherein, the Laplace equation is transformed into ordinary differential equations. This necessitates that the boundary be a coordinate surface for one of the special orthogonal coordinate systems for which the separation of variables is possible. For arbitrary boundaries, however, any analytical formulation will in the end, inevitably, become numerical. Several methods of solving this elliptic PDE (partial differential equation) are presented in [4] and [23]. Most of these methods employ

finite difference schemes or a form of Green's function. In some two-dimensional cases the direct problem of potential flow can be solved through what is known as the conformal transformation methods. These methods are based on finding an appropriate conformal transformation that transforms the boundary curve of the body into a curve for which the solution is known. They are, however, unsuitable for the computation of the flow about several bodies.

The method of solution which appears to be the most efficient for the application in a fluid flow, and which was consequently chosen, is known as the panel method [16], [27]. This method is based on replacing the required solution to the Laplace equation over the given domain with a surface integral. The advantage of using the panel method is in the fact that the pressure distribution over the solid boundary can be obtained directly, without the necessity of solving for the flow field throughout the entire domain.

The implementation of the panel method is presented in the next section.

Once the Laplace equation is solved for the potential function ϕ over the entire domain the velocity vector can be computed from Equation (5.7). The pressures are then determined from Equation (5.13).

5.4 PROPOSED METHOD OF FLOW COMPUTATION

In this section we will establish a numerical methodology for the solution of the Laplace equation, Equation (5.12), which, with the boundary condition (5.14), is governing the flow of an incompressible and irrotational fluid. We will also assume that the flow is two-dimensional and steady. Although, the methodology for a general three-dimensional case is analogous. Therefore in the present analysis the domain over which the solution is sought reduces to a region within the x - y plane and the solid boundary becomes a line in this plane. A typical situation is shown in Figure 5.1.

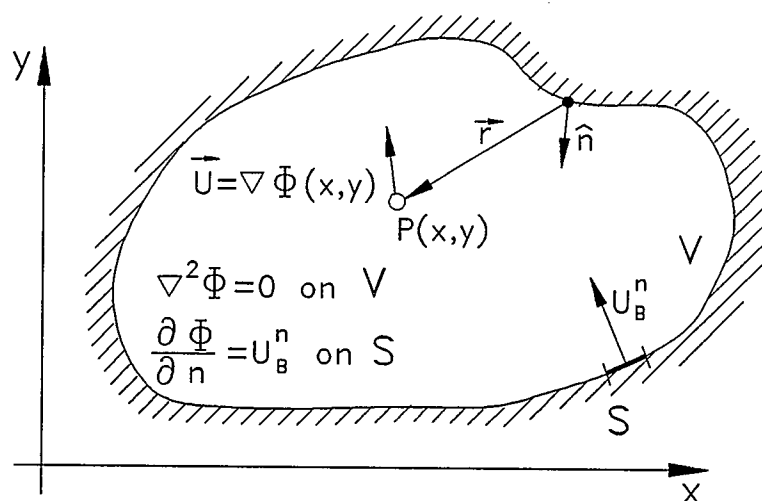


Figure 5.1. A schematic diagram of a physical domain for the Laplace equation.

The solution to the Laplace equation is obtained by what is known as the panel method [16],[27]. In this method the

solution to the equation over the domain is replaced with the surface integral. Consequently the problem is reduced to the solution of a system of linear algebraic equations for the strengths of the sources distributed on the solid boundary.

If ϕ represents a scalar function that satisfies a two-dimensional Laplace equation,

$$\nabla^2 \phi = \frac{\partial^2 \phi}{\partial x^2} + \frac{\partial^2 \phi}{\partial y^2} = 0 \quad (5.16)$$

then it can be shown (see, for example, [4], [25]) that at any given point $P(x,y)$ within the domain, the scalar function ϕ of Equation (5.16) is given by:

$$\phi(x,y) = \frac{1}{2\pi} \oint_s \left[\ln(r) \frac{\partial \phi}{\partial n} - \phi \frac{\partial \ln(r)}{\partial n} \right] ds \quad (5.17)$$

The first term in square brackets in Equation (5.17) corresponds to the Neumann problem for which $\partial \phi / \partial n$ is prescribed on the boundary (of interest here). The second term in the brackets, together with Equation (5.16), comprises a Dirichlet boundary value problem, for which ϕ is specified on the boundary. In fluid flow these two terms correspond to the contributions made to ϕ by sources and doublets distributed on the solid boundary, respectively. Hence, we may write

$$\phi(x, y) = \frac{1}{2\pi} \oint_S \sigma \ln(r) dS \quad (5.18)$$

where σ is interpreted as a surface source distribution per unit length.

If the solution to the Laplace equation (5.16), with a Neumann condition (5.14) prescribed on the boundary, is sought over the given domain on which a flow having a potential ϕ_∞ is superimposed, then this potential is included in Equation (5.18) and

$$\phi(x, y) = \phi_\infty(x, y) + \frac{1}{2\pi} \oint_S \sigma \ln(r) dS \quad (5.19)$$

The line source distribution σ is obtained from the boundary condition on S :

$$\frac{\partial \phi}{\partial n} = \nabla \phi \cdot \hat{n} = \vec{u} \cdot \hat{n} = u_B^n \quad (5.20)$$

where u_B^n is the normal velocity component of the rigid boundary. Clearly, if the boundary is stationary then $u_B^n = 0$.

Let us now consider a problem of a steady potential flow around an arbitrary body. The flow is governed by the potential function of Equation (5.19), where $\phi_\infty(x, y)$ represents the potential of the free (undisturbed) stream, r is the distance from point $P(x, y)$ to the given point on the

boundary, and σ is the unknown source strength distribution function. The source distribution function can be established by replacing the surface of the body with small line segments or panels over which σ can be considered constant, as shown in Figure 5.2.

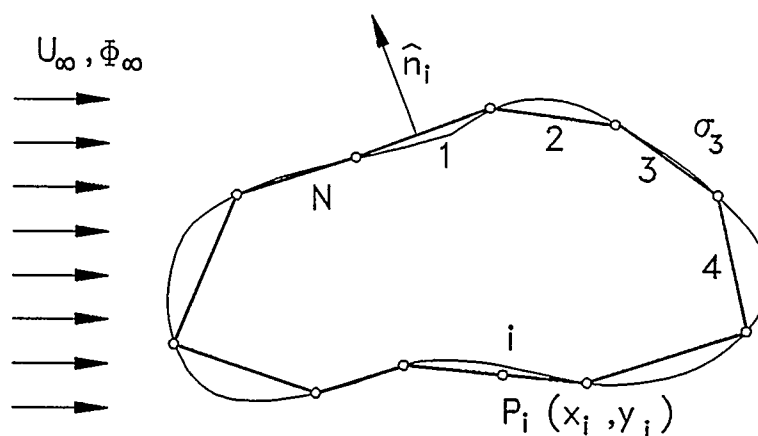


Figure 5.2. A panel representation of a body.

The strength of each source panel is chosen in such a way that the resultant global velocity field satisfies the boundary conditions at each panel. Namely, that the relative velocity component in the direction normal to that panel is zero. If N represents the number of panels used to describe the rigid body then the total potential at a given external point is the sum of contributions from each panel. Hence Equation (5.19) may be rewritten as

$$\phi(x, y) = \phi_{\infty}(x, y) + \frac{1}{2\pi} \sum_{j=1}^N \sigma_j \int_j \ln(r) dS \quad (5.21)$$

where σ_j represents the strength of the source distributed over panel j . The line integral in the above equation can be thought of as a potential generated by a unit strength source distributed on panel j and for the simplicity of the equations we will denote it as ϕ_j . Equation (5.21) becomes

$$\phi = \phi_{\infty} + \frac{1}{2\pi} \sum_{j=1}^N \sigma_j \phi_j \quad (5.22)$$

A control point is chosen for each panel at which the boundary condition must be satisfied. Hence, we may write N boundary conditions (one for each of the N panels). By Equation (5.20) we have

$$\frac{1}{2\pi} \sum_{j=1}^N \sigma_j \nabla \phi_j^i \cdot \hat{n}_i = -\nabla \phi_{\infty}^i \cdot \hat{n}_i + u_i^n, \quad (i=1, N) \quad (5.23)$$

where $\frac{1}{2\pi} \phi_j^i = \frac{1}{2\pi} \phi_j(x_i, y_i)$ represents the potential induced by the unit strength panel j on the control point of panel i , and $\hat{n}_i$ is the unit outward normal vector of panel i . Also, by Equation (5.7) we let $\vec{u}_j^i = \frac{1}{2\pi} \nabla \phi_j^i$ denote the velocity induced by unit strength panel j on the control point of panel i . Equation (5.23) can now be conveniently written as

$$\sum_{j=1}^N a_{ij} \sigma_j = -\vec{u}_{\infty}^i \cdot \hat{n}_i + u_i^n, \quad (i=1, N) \quad (5.24)$$

where $a_{ij} = \vec{u}_j^i \cdot \hat{n}_i$ is referred to as the influence coefficient of panel j on panel i . The terms on the left hand side of Equation (5.24) are known, since they depend only on the free stream velocity field and the motion of the rigid boundaries. Herein the potential flow problem was reduced to solving the above system of N linear algebraic equations for N unknown strengths of source panels. The influence coefficients a_{ij} depend only on the known geometry of the body under consideration, and therefore, can always be determined.

Consider a line segment of a uniformly distributed source of unit strength and having length (d), (Figure 5.3).

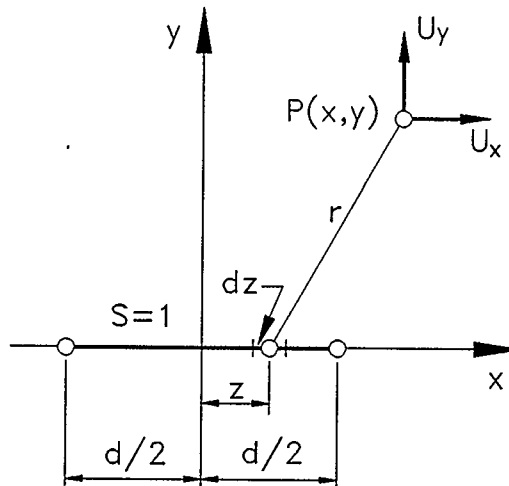


Figure 5.3. A diagram for calculating a velocity induced by a source line segment.

The velocity induced by an infinitesimal section of the segment (dz) on point $P(x, y)$ is:

$$du_x = \frac{1}{2\pi} \frac{x - z}{(x - z)^2 + y^2} dz \quad (5.25a)$$

$$du_y = \frac{1}{2\pi} \frac{y}{(x - z)^2 + y^2} dz \quad (5.25b)$$

Integrating Equations (5.25) over z from $(-d/2)$ to $(d/2)$ we obtain the expressions for the velocity components:

$$u_x = \frac{1}{4\pi} \ln \left[\frac{(x + d/2)^2 + y^2}{(x - d/2)^2 + y^2} \right] \quad (5.26a)$$

$$u_y = \frac{1}{2\pi} \left[\tan^{-1} \left(\frac{x + d/2}{y} \right) - \tan^{-1} \left(\frac{x - d/2}{y} \right) \right] \quad (5.26b)$$

We recall that an influence coefficient, a_{ij} , is a dot product of an outer normal vector of panel i , $\hat{n}_i$, and a vector of velocity, $\vec{u}_j^i$, induced at the control point of panel i , $P_i(x_i, y_i)$, by a uniformly distributed source of a unit strength at panel j .

$$a_{ij} = \vec{u}_j^i \cdot \hat{n}_i \quad (5.27)$$

It can be easily demonstrated that the dot product of two vectors in cartesian coordinates does not depend on the choice of the origin or the orientation (rotation) of the system of coordinates. Consequently, the influence coefficients can be evaluated in a conveniently chosen local system of coordinates. The natural choice for the local system of

coordinates seems to be the one for which the origin is located at the mid point of the source panel and for which the x-axis coincides with the panel itself, as shown in Figure 5.4.

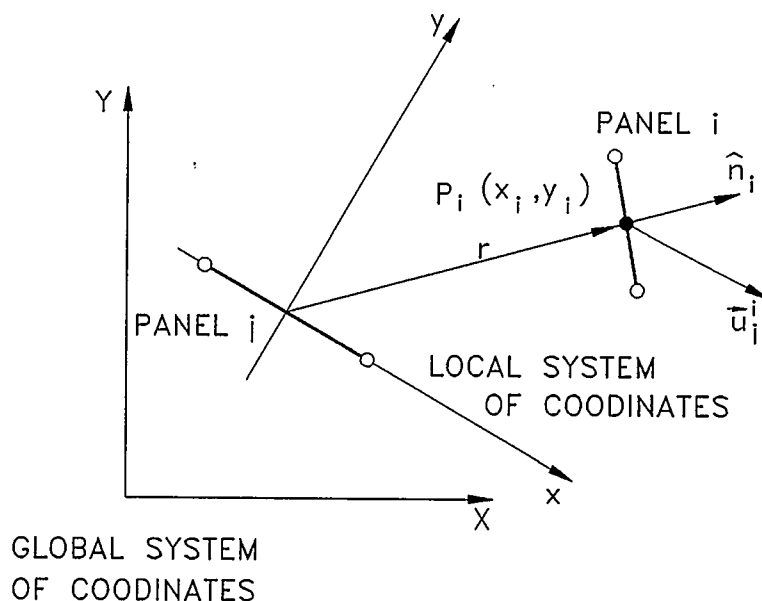


Figure 5.4. A schematic diagram for calculation of the influence coefficients.

We will use upper case letters to denote the global system of coordinates and lower case letters for the local system of coordinates.

In the local system of coordinates the components of the velocity induced at the control point of some panel i , by panel j , are readily computed from Equation (5.26a) and (5.26b), where $x = x_i$, $y = y_i$, are the coordinates of the control point of panel i expressed in the local system of

coordinates. We may therefore write Equation (5.27) in terms of coordinate components as

$$a_{ij} = u_{xj}^i n_{xi} + u_{yj}^i n_{yi} \quad (5.28)$$

where n_{xi} , n_{yi} , are the local components of the normal vector of panel i . These components are known in the global system of coordinates and can be expressed in the local coordinate system through the following transformation:

$$n_{xi} = n_{yj} n_{xi} - n_{xj} n_{yi} \quad (5.29a)$$

$$n_{yi} = n_{xj} n_{xi} + n_{yj} n_{yi} \quad (5.29b)$$

where n_{xj} , n_{yj} , n_{xi} , n_{yi} , are, respectively, the known components of outward unit normal vectors of panels j and i expressed in the global system of coordinates.

In a similar way the local coordinates of the control point on panel i can be determined:

$$x_i = n_{yj} (X_i - X_j) - n_{xj} (Y_i - Y_j) \quad (5.30a)$$

$$y_i = n_{xj} (X_i - X_j) + n_{yj} (Y_i - Y_j) \quad (5.30b)$$

This concludes the computation of the influence coefficients. The system of linear algebraic equations (5.24) can now be solved for the strength of panel sources σ_j .

The flow velocity vector at any point $P(X, Y)$ can now be calculated according to:

$$\vec{u}(X, Y) = \vec{u}_\infty(X, Y) + \sigma_j \vec{u}_j(X, Y) \quad (5.31)$$

where $\vec{u}_j(X, Y)$ denotes the velocity vector induced by a unit strength source panel j . The components of this vector are evaluated first in the local system of coordinates using Equation (5.26a) and Equation (5.26b) and then are transformed in to the global system of coordinates by applying an inverse transformation to the one used in Equation (5.29).

Finally, the pressure at any point in the flow is determined from Equation (5.13).

$$p(X, Y) = p_\infty + \frac{1}{2} \rho u_\infty^2 - \frac{1}{2} \rho [u(X, Y)]^2 \quad (5.32)$$

This concludes the analytical formulation of a problem of two-dimensional potential flow around an arbitrary body. The appropriate computer routines were programmed and tested. The system of linear algebraic equations (5.24) is, at present, solved using a standard Gaussian elimination method with backward substitution. The advantage of using this method is in simplicity of its application. However, the method is not as fast as some others particularly for a large number of equations. Approximately N^3 multiplications are required, as

well as, N^2 additions and divisions. Consequently, the accumulation of round-off errors through the many algebraic operations can cause the accuracy to deteriorate if N is large. At present, this does not seem to constitute a problem. Should, however, such a situation arise, the Gaussian elimination can be substituted with one of the iterative methods, for example, Gauss-Seidel or SOR (successive over-relaxation), which allow for any desired accuracy of computations [7]. Regrettably, the iterative procedures require the introduction of an initial guess which proximity to the actual solution will affect the speed of computation. Incidentally, such a guess could be provided by the standard Gaussian elimination method.

The method of validation of the presently used computer routines for flow calculation will be presented briefly in the next section of this chapter.

5.5 VERIFICATION OF THE PROPOSED METHOD OF FLOW COMPUTATION

In order to verify the methodology of potential flow computation developed in this chapter, a velocity field around a certain body was calculated and compared with the exact analytical solution.

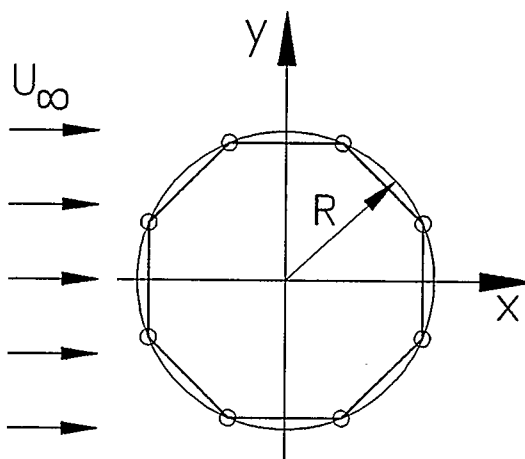


Figure 5.5. A panel representation for the cylinder.

As a test for the external flow a stationary cylinder of radius R was considered (in two dimensions the cylinder reduces to a circle), in an onset flow parallel to the x -axis having a free stream velocity u_∞ . The cylinder was represented by eight panels as shown in Figure 5.5. The analytic solution to the problem was obtained by placing a doublet at the center of the circle. The strength of the doublet was determined from the boundary condition that

$u_x(x=-R, y=0) = 0$. The resultant analytical description is given by the following equations.

$$\phi(x, y) = u_{\infty} \left[x + \frac{R^2 x}{x^2 + y^2} \right] \quad (5.33')$$

and

$$u_x = u_{\infty} \left[1 + \frac{R^2 (y^2 - x^2)}{(x^2 + y^2)^2} \right] \quad (5.34a)$$

$$u_y = -u_{\infty} \frac{2R^2 xy}{(x^2 + y^2)^2} \quad (5.34b)$$

The computed flow velocities agreed well with those determined analytically, even though a relatively low number of panels (8) were used.

The applicability of the computational methodology to the channel flow was verified using the example shown in Figure 5.6. The figure shows a channel with an "L" bend and the corresponding panel representation. The velocity profile in the cross-section marked A-A was computed and compared with the analytical solution.

Since a fluid flow around the bend shown in Figure 5.6 is potential it must be irrotational. Such motion would then be equivalent to a free, irrotational vortex. Hence, the analytical solution is obtained from the appropriate governing equations. We have:

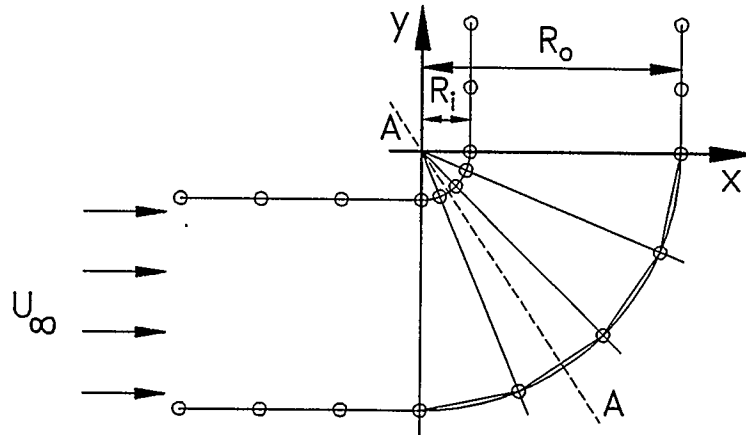


Figure 5.6. Panel representation of a "L" shaped channel.

$$u_x = -u_\infty \frac{y(R_o - R_i)}{(x^2 + y^2) \ln(R_o/R_i)} \quad (5.35a)$$

$$u_y = u_\infty \frac{x(R_o - R_i)}{(x^2 + y^2) \ln(R_o/R_i)} \quad (5.35b)$$

The computed and analytical velocity fields were found to be in good agreement.

The examples considered added strength to the argument of applicability of this particular method of potential flow calculation in both external and channel flows. It cannot, however, be guaranteed that good results will be obtained for any shape of the boundaries. Practice shows, for example, that problems may be encountered when the flow around a body with sharp concave corners is being computed. For such systems a considerable "leakage" through the solid boundary

may be encountered. The "leakage" occurs because the source panel is of some finite length. The boundary condition is satisfied at the control point, but may not necessarily be satisfied at any other point on the panel. Consequently there may exist a net flow through the boundary. The problem of leakage may be corrected by introducing more panels to smooth out the sharp concave corners or by using a variable source distribution over the panels rather than a uniform one, and by applying the boundary conditions to appropriately more control points at each panel. One could also incorporate an integration scheme into the procedure of calculating the strengths of the panel sources, whereby, the boundary conditions would reflect zero net flow over the entire element, rather than a zero relative velocity component at the specified control point of the panel.

5.6 CONCLUSIONS

The method of flow determination presented in this chapter has been successfully implemented within the model. The method has been found applicable to both the external and channel flows.

The model's description of the solid boundaries using straight line segments is directly compatible with the definition of source panels used for flow computation. Hence, the present routine can be integrated in the model straight away, without any need of redefining the boundaries.

In terms of the numerical computation, the primary goal in establishing the flow field is the determination of the source panel strengths. The strengths of the source panels which are given by the system of simultaneous algebraic equations, Equation (5.24), are successfully computed by this analysis.

No significant "leakage" through the solid boundaries was found for a variety of different systems tested. And, the methods of treating this problem, should it arise, are briefly discussed in Section 5.5.

It is believed that for a vast majority of systems for which the present model is intended, the potential flow theory constitutes an adequate tool for flow determination.

It should also be stated that at present the effects of the solid bodies on the fluid flow are neglected. This, in terms of the simulation process, can be partially corrected with modifications to the drag coefficients used in determining the forces acting on the solid bodies.

CHAPTER 6

FORCES ON RIGID BODIES

6.1 INTRODUCTION

Two primary types of forces that the rigid bodies present in the system experience, are the external forces which depend on the particular process that the multi-body system is undergoing, and the internal forces which are the result of the interactions among the individual bodies.

The character of the external forces for a particular process must be identified, and it comprises a necessary input for the simulation program. This is discussed in Section 6.2.

The internal forces include the contact, friction, and impact forces. The contact forces deserve special mention, and these are discussed in Chapter 7. The friction and impact forces are considered in this chapter in Section 6.3.

6.2 EXTERNAL FORCES

The external forces represent a driving mechanism for the motion of multi-body systems, and thus must be supplied to the simulation program as an input. In the simulation program a special routine is allocated in which the character of the external forces acting on the system is pre-programmed. The routine communicates the information about the forces to the main simulation program. The routine must be programmed in such a way that for every rigid body in the system the x and y components of the total external force can be determined from the position of the body's center, its velocity, and its acceleration and mass. The simple rigid bodies which are represented by disks are treated like particles, and the external forces are applied to their respective centers. For the complex rigid bodies or quasi rigid body sub-systems, the external forces are calculated for every disk comprising them and applied to its center.

The external forces and their character will differ depending on the process under consideration and thus cannot be described in a general sense. Two specific cases will however be discussed next.

Gravitational Force. The gravitational force acting on a given disk j is given by:

$$\vec{F}_j = m_j \vec{g} \quad (6.1)$$

where $\vec{g}$ is the gravitational acceleration. The components of $\vec{g}$ for a particular choice of the inertial frame of reference must be known.

Drag forces. The drag forces must be determined when a motion of a multi-body system in a fluid media is under consideration. In general a drag force on a solid body is proportional to the square of the relative velocity of the body with respect to the fluid. Consequently, the computation of the drag forces requires the knowledge of the fluid velocity field. This information can be either supplied as an input for a particular system or generated by the model (Chapter 5).

There are two types of drag force: a form drag and a skin friction drag. Both types can be described by one general relation. Namely, if $\vec{u}_B$ denotes the velocity of the center of a disk and $\vec{u}_F$, to be the velocity of the fluid at that point then the drag force on the disk is given by

$$\vec{F} = -\frac{1}{2} \rho A_{eff} C_D (\vec{u}_B - \vec{u}_F) |\vec{u}_B - \vec{u}_F| \quad (6.2)$$

where A_{eff} is the effective projected area and C_D the drag coefficient. For the skin friction drag A_{eff} represents the area of the body's surface exposed to the fluid. In the case of the form drag A_{eff} is the maximum cross-sectional area of the body normal to the direction of the mean flow. The coefficient of drag C_D for skin friction depends mostly on the roughness of the solid surface and the viscosity of the fluid involved and have to be determined experimentally. For the form drag C_D depends on the shape of the body, and similarly to skin friction drag coefficient, is determined experimentally. Drag coefficients can be frequently found tabulated for various shapes in textbooks on fluid mechanics (see [6] for example).

6.3 INTERNAL FORCES

The two types of internal forces that are of interest here are the frictional and impact forces. The frictional forces result from the relative motion of bodies in contact and depend on the unknown reaction (constraint) forces. Dry friction is only considered here. It is assumed that the frictional force is proportional to the reaction force (Coulomb Law of friction).

The impact loads result from the collisions among the rigid bodies. It is assumed that a collision occurs every time a disk event (Chapter 2) takes place. The general collision theory is used in the Lagrangian form to model impacts. The impact forces must be evaluated as these may significantly affect the contact forces among the bodies, and thus cause a force event. As a final assumption it should also be stated that the collision theory used assumes no friction.

The following two sub-sections present the formulation for the evaluation of the frictional and impact forces used in the present model.

6.3.1 Frictional Forces

In the general case of modelling a system of rigid bodies the friction can be handled by simply modifying the generalized forces. Similarly to Equation (3.13) we could write an expression for the virtual work done by the frictional forces, δW_v . Namely, for a system of particles,

$$\delta W_v = \sum_{\alpha=1}^d \lambda_{\alpha} \left[\sum_{i=1}^n v_{\alpha i} \delta q_i \right] \quad (6.3)$$

where $v_{\alpha i}$ is the generalized frictional matrix given by

$$v_{\alpha i} = \sum_{j=1}^N \left[\sum_{p,r,s} \epsilon_{prs} \mu_j^r \frac{\partial f_{\alpha}}{\partial x_j^s} \frac{\partial x_j^p}{\partial q_i} \right] \quad (6.4)$$

where superscripts p, r, s denote the appropriate Cartesian coordinates, ϵ_{prs} is the permutation constant, and μ^r represents the cartesian components of the vector orthogonal to both the constraint force and the relative velocity vector, whose length is equal to the coefficient of friction.

The equations of motion (3.17) for a system of particles, or Equation (3.24) for rigid bodies, with friction included can now be written as

$$[A] \cdot [\ddot{q}] + [B] \cdot [\dot{q}] = [Q] + [G]^T \cdot [\lambda] + [\Phi]^T \cdot [\lambda] \quad (6.5)$$

Coefficients $v_{\alpha i}$ of matrix $[\Phi]$ are in general, non-linear with respect to $\dot{q}$ and q . Consequently, the equations of (6.5) are highly non-linear and cannot be, except for some very simple cases, solved without a grave numerical difficulty, that is, if the equations can be generated at all.

It is common practice in modelling of multi-body systems [28] that during the time simulation the frictional forces calculated in the preceding time step are entered into the equations of motion (6.5) as known parameters. This is, of course, analogous to modification of the generalized forces. Namely, we can write symbolically,

$$[Q]_{t+\Delta t}^* = [Q]_{t+\Delta t} + [\Phi]_t$$

The equations (6.5) now become identical to equations given by (3.17) where the vector of the generalized forces is given by $[Q]^*$. The modified equations can be solved, and upon the solution new frictional forces can be computed for the next time increment. In other words, the frictional forces "lag" the constraint forces by a time step. This suggests that an error is introduced systematically during the simulation. However, this error is of second order (acceleration) and is not considered significant. This approach is also used in our

model.

The above reasoning suggests that since the frictional forces are assumed known at every time step, they can be simply evaluated in the cartesian coordinate system and then added to the active forces of the system from which, by the application of the principle of virtual work, the modified generalized forces can be computed. This can be explained by considering a general model for friction as illustrated in Figure 6.1. For the clarity of reasoning the sphere is representing a rigid body.

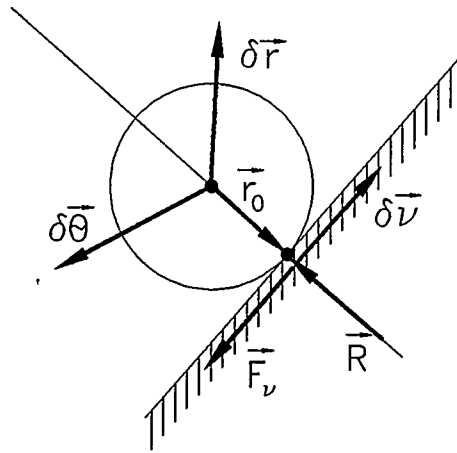


Figure 6.1. An illustration of the frictional forces.

In the figure $\delta\vec{\theta}$ is the vector of angular virtual displacements, $\delta\vec{v}$ is the total virtual displacement across the constraint, $\vec{r}_0$ represents a vector from some chosen point within the body (center of the sphere) to the point of interface on the body's surface, and $\vec{F}_v$ is the frictional

force at the constraint. The other interacting body is represented by a stationary plane. This does not reduce the generality of the model, since the indicated displacements can be thought of as the relative displacements of one body with respect to the other.

The virtual work done by the friction in the constraint can be written as

$$\delta W_v = \vec{F}_v \cdot \delta \vec{v} \quad (6.6)$$

The virtual displacement across the constraint can be calculated as

$$\delta \vec{v} = \delta \vec{r} + \delta \vec{\theta} \times \vec{r}_0$$

and hence Equation (6.6) is rewritten as

$$\delta W_v = (\delta \vec{r} + \delta \vec{\theta} \times \vec{r}_0) \cdot \vec{F}_v$$

Remembering that a dot product is commutative and the triple product does not depend on the order of components as long as the cyclic permutation with respect to the cross product is maintained, we can write the above relation as:

$$\delta W_v = \vec{F}_v \cdot \delta \vec{r} + \vec{r}_0 \times \vec{F}_v \cdot \delta \vec{\theta} \quad (6.7)$$

The second term in the above equation represents a moment created by the frictional force about point "O". We write,

$$\delta W_v = \vec{F}_v \cdot \delta \vec{r} + \vec{M}_v \cdot \delta \vec{\theta}$$

which is directly compatible with the virtual work principle for a system of rigid bodies, Equation (3.18). Consequently, friction can be handled by simply adding the corresponding frictional forces and frictional moments to the active loads and proceeding with setting up the equations of motion as in (Section 3.5) with new modified loads.

The equivalence of our method with the general procedure, Equation (6.5) can be demonstrated as follows. For a system of N particles we have,

$$\delta W_v = \sum_{j=1}^N \vec{F}_{v_j} \cdot \delta \vec{r}_j$$

but, $\vec{F}_{v_j} = \vec{\mu}_j \times \vec{R}_j$. Also repeating the procedure of Section 3.2 to obtain an expression for virtual work in terms of generalized coordinates, and remembering that

$$\delta \vec{r}_j = \sum_{i=1}^n \frac{\partial \vec{r}_j}{\partial q_i} \delta q_i, \quad \vec{R}_j = \sum_{\alpha=1}^d \lambda_{\alpha} \nabla_j f_{\alpha} \quad (6.8)$$

we obtain

$$\delta W_v = \sum_{j=1}^N \left[\sum_{\alpha=1}^d \lambda_{\alpha} \vec{\mu}_j \times \nabla_j f_{\alpha} \right] \cdot \left[\sum_{i=1}^n \frac{\partial \vec{r}_j}{\partial q_i} \delta q_i \right] \quad (6.9)$$

Performing the triple product of Equation (6.9), and rearranging the terms Equation (6.3) can be obtained.

To complete the proposed friction handling method we need to evaluate the vector of frictional force. This is equivalent to the determination of the vector $\vec{\mu}_j$. For this reason we will define two vectors $\hat{n}_{R_j}$, $\hat{n}_{U_j}$ which represent unit vectors in the direction of the constraint force and the relative velocity at the constraint of body j . The two vectors are given by

$$\hat{n}_{R_j} = \frac{\vec{R}_j}{\sqrt{\vec{R}_j \cdot \vec{R}_j}} \quad , \quad \hat{n}_{U_j} = \frac{\vec{u}_j}{\sqrt{\vec{u}_j \cdot \vec{u}_j}}$$

where $\vec{u}_j$ is the vector of relative velocity. Vector $\vec{\mu}_j$ can now be expressed as

$$\vec{\mu}_j = c_f \hat{n}_{U_j} \times \hat{n}_{R_j} \quad (6.10)$$

where c_f is the coefficient of friction. The frictional force which must be orthogonal to the constraint (contact) force $\vec{R}_j$ is thus given by

$$\vec{F}_{v_j} = \vec{\mu}_j \times \vec{R}_j \quad (6.11)$$

The frictional forces on every body in the system can now be calculated. The corresponding loads, as identified by Equation (6.7) can be determined and added to the active loads (forces and moments) acting on the body. The resultant equations of motion are represented by Equation (3.24) with the modified vector of the generalized forces.

As a final note we should again point out that our choice of generalized coordinates (Section 3.6), the utilization of the concept of quasi rigid body subsystems (Section 2.3.2), and proposed methodology of event determination and handling (Section 2.3.3), was aimed at reducing the number of the algebraic constraint equations in the overall system of motion equations. Therefore, in effect, not all the constraints are represented by the corresponding constraint equations, and hence, not all constraint (contact) forces $\vec{R}_j$ can be evaluated through the use of Lagrange multipliers as given by Equation (6.8). Thereafter the contact forces are evaluated by a different method. This will be described in Chapter 7.

6.3.2 Impact Forces

Every time a disk event is detected, it is assumed that a collision occurred. A collision can be described by a Newton's collision rule.

$$(\vec{v}_{n_{i/j}})_1 = -\epsilon \cdot (\vec{v}_{n_{i/j}})_0 \quad (6.12)$$

where ϵ is the coefficient of restitution, the subscript $n_{i/j}$ indicates a normal component of the velocity of disk i relative to disk j , and the subscripts 0 and 1 refer to times just prior to and right after the collision, respectively. Alternately, Equation (6.12) could be written as,

$$(\vec{v}_{n_i} - \vec{v}_{n_j})_1 = -\epsilon \cdot (\vec{v}_{n_i} - \vec{v}_{n_j})_0 \quad (6.13)$$

The normal direction is identified as being along the line connecting the centers of the colliding disks.

The change of sign in Equation (6.12) suggests that during the collision, there is an instant in time at which the relative velocity becomes zero. This instant is referred to as the point of maximum compression, and it will be denoted with a subscript "c". It is assumed that during the collision the bodies involved deform (compress) up until this

point. This is followed by the process of restitution during which the shapes of the bodies return to their original form and kinetic energy which was stored as the strain energy, is returned (in whole or in part) to the bodies. At the point of maximum compression we thus have,

$$(\vec{v}_{n_{i/j}})_c = 0 \rightarrow (\vec{v}_{n_i})_c = (\vec{v}_{n_j})_c \quad (6.14)$$

With the use of Equation (6.14), Newton's Law of collision, Equation (6.13), can be written in the form of the following two conditions:

$$\left. \begin{aligned} (\vec{v}_{n_i})_1 - (\vec{v}_{n_i})_c &= -\epsilon \cdot [(\vec{v}_{n_i})_0 - (\vec{v}_{n_i})_c] \\ (\vec{v}_{n_j})_1 - (\vec{v}_{n_j})_c &= -\epsilon \cdot [(\vec{v}_{n_j})_0 - (\vec{v}_{n_j})_c] \end{aligned} \right\} \quad (6.15)$$

Since, normal components of the velocities before collision (0) are known, then the normal velocity components after the collision (1) can be evaluated from Equation (6.15), provided that the normal velocity components at the point of maximum compression (c) can be determined.

For a system of N disks which are in contact the general theory of collisions used in the present model assumes that the point of maximum compression is reached by all disk pairs at the same time. Hence, we can introduce the so-called

impulse forces $\mathcal{F}$ and apply a conservation of momentum principle to each disk from the initial state (before the collision) to the point of maximum compression. We have

$$m_i(\vec{v}_i)_c - m_i(\vec{v}_i)_0 = \sum_j \vec{\mathcal{F}}_{ij}, \quad (i=1, N) \quad (6.16)$$

where the subscript j denotes all disks which are in contact with the given disk i . The orientations of the impulse forces are along the lines connecting the centers of the respective disks, and thus are known. Hence, Equation (6.16) represents a system of $2N$ with $2N + N_c$ unknowns, $2N$ velocity components and N_c magnitudes of the impulse forces, where N_c is the number of the points of contact in the disk cluster. However, at the point of maximum compression we additionally have N_c conditions of Equation (6.14) (one for every connection). The system of equations (6.16) together with a set of conditions (6.14) can be solved for the velocity components at the point of maximum compression. Thereafter, the normal velocity components at this point can be calculated. Subsequent use of Equation (6.15) will yield the velocities of the rigid bodies at the instant in time after the collisions.

Finally, if during the solution of Equation (6.16) one of the impulse forces $\mathcal{F}$ becomes positive (tensile) then the appropriate connection is deleted from the system, as such a situation cannot arise for a system with one-sided

constraints.

The methodology for handling the collisions among the rigid bodies was developed in [31]. The method of generation of the appropriate collision equations and its numerical implementation can be found in that reference. In this section the general theory was briefly presented for completeness.

6.4 CONCLUSIONS

In this chapter various external and internal forces that the rigid bodies experience were identified and discussed. In Section (6.2) it was demonstrated that the drag forces acting on the rigid bodies in a fluid media can be estimated in the present model. The computer routine that computes the fluid velocity field (Chapter 5) is based on the potential flow theory. At present, the influence of the solid bodies on the fluid flow is not considered.

The method of handling friction among the rigid bodies was presented in Section 6.3.1. It was proposed that at each time increment the frictional forces be calculated after the equations of motion (modified by frictional forces from the previous time step) are integrated. This technique allowed us to consider friction which otherwise could not be considered. This, however, led to the introduction of a second order error.

The method of handling impacts among the rigid bodies, which was developed in [31] was presented here for completeness. This method does not consider friction, as the tangential components of velocity are assumed unchanged during collisions.

CHAPTER 7

CONTACT FORCES AMONG THE RIGID BODIES AND OBSTRUCTIONS

7.1 INTRODUCTION

Contact forces among the rigid bodies and obstruction line segments result from the external forces acting on the rigid bodies, and are the forces which must be applied to the respective bodies at the points of contact in order for the entire configuration to move in the same manner that it would if the rigid bodies were jointed at these points. For example, let us consider a pair of disks in contact. There are some external forces applied to each disk. We can replace the specified connection between the two disks with two forces of equal magnitude and opposite direction (no additional net force on the system) applied to each body at the point of contact. These two forces can now be considered as external forces. Consequently, the pair of disks is replaced by two separate disks which will behave as if the two disks were connected.

The contact forces are said to be tensile if they are directed away from the point of contact, and compressive otherwise. Figure (7.1) shows an example of tensile and

compressive contact forces. The tensile contact forces are assumed positive, while the compressive contact forces will have a negative sign. This convention will be used throughout this chapter.

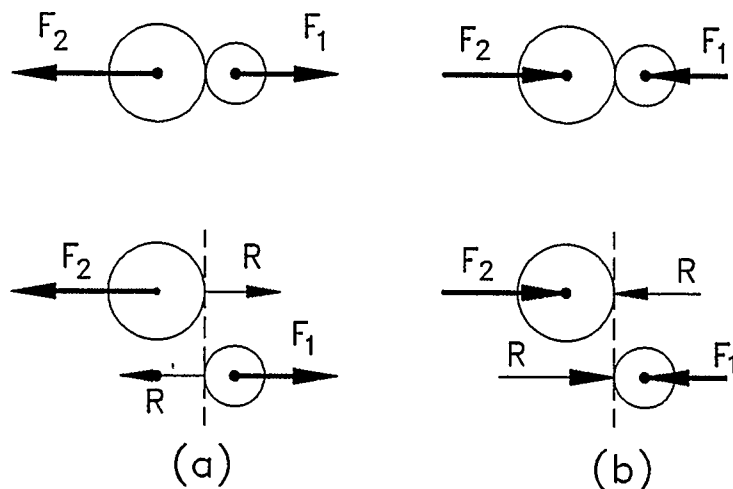


Figure 7.1. An example of tensile (a) and compressive (b) contact forces.

At each point in time during the simulation procedure the contact forces among the rigid bodies and obstruction line segments must be known. This is because the connectivities among the rigid bodies and obstruction line segments are maintained during the simulation process until the contact forces between the given bodies become tensile and greater than some pre-specified value. For example, if a motion of a cohesionless granular material is to be simulated then whenever a contact force greater than zero (tension) is

encountered, the corresponding connection is deleted. Similarly, if the cohesion between the rigid bodies is present, then the given connection is deleted whenever the corresponding contact force is greater than the cohesive strength.

The contact forces are evaluated not only among the particular rigid bodies but also among the disks composing the complex bodies present in the system (Chapter 4). This is done in order to be able to simulate the processes in which a break-up of solids may occur, for example, an interaction of moving ice with a structure. In such an instant, a break-up of a rigid body takes place whenever the contact forces between the component disks, along some line through the body are greater than the strength of the material. Clearly, the simple rigid bodies, i.e. those composed of only a single disk, cannot be broken.

The following chapter will deal with the evaluation of the contact forces among all the disks present in the system. This, as mentioned above, includes both the disks representing simple bodies and those which are a part of the complex bodies.

Since our model is two-dimensional, then for every disk present in the system there are two corresponding equations

representing Newton's Second Law of motion. These equations correlate the total acceleration of the disk to the net force on it, including the contact forces. Consequently, for any conglomerate of disks for which the number of points of contact is less than or equal to the number of available dynamic equilibrium equations, i.e. twice the number of disks, the unknown contact forces can be evaluated using these equations. This will be referred to as the method of dynamic equilibrium.

It may however occur that the number of points of contact in a given disk conglomerate exceeds the number of equations of motion, and the dynamic equilibrium cannot be used to evaluate the contact forces. In this case it is proposed that the entire conglomerate be replaced by a truss system. In such a system a truss would represent two disks in contact, with its endpoints located at the centers of the respective disks. Clearly, the centers of the disks would become nodes at which the external forces, including the inertial acceleration terms, are applied. The entire structure can then be solved from static equilibrium for the forces in the trusses which, of course, are the unknown contact forces among the disks. This method is referred to as the method of static equilibrium.

The two methods of determination of the contact forces are presented in this chapter.

7.2 DYNAMIC EQUILIBRIUM ANALYSIS

If a disk is in contact with one or two other entities (those could be other disks or the obstruction line segments), then the contact forces between a given disk and the entities can be calculated from the dynamic equilibrium equations. These equations are based on Newton's Second Law of motion. The law states that the sum of all the forces in a given direction is equal to the mass of the body multiplied by its acceleration in that direction. This can be written as:

$$\sum F_{\xi} = m a_{\xi}$$

where ξ indicates a given direction. Since our model is two-dimensional, we can write two such equations for a disk, in two distinct directions. Hence, the limitation that the given disk has no more than two points of contact.

We will now present the method of utilizing the above equation to calculate the contact forces for disks having one or two points of contact. The contact forces will be denoted as F_c . A contact force between two disks is directed along the line connecting their centers, while a contact force between a disk and an obstruction line segment is along the line connecting the center of a disk with the corresponding

point of contact.

The computation of contact forces using this method is done in the following way. The given disk conglomerate is searched for disks with a one or two points of contact. Once such a disk is identified the contact forces acting on it are computed from the dynamic equilibrium equations. The disk is then, temporarily deleted from the conglomeration, and in its place, the computed contact forces are applied to the entities that it contacted, as external forces. The remaining disks are then again scanned for the number of points of contact, and the ones with one or two points have the corresponding contact forces determined. The procedure is repeated until each of the remaining disks have at least three points of contact. At such an instant the dynamic equilibrium cannot be used for evaluation of the contact forces, and the method of static analysis Section 7.3 is then used.

7.2.1 Disks with One Point of Contact

Let us assume that disk i is in contact with disk, or a line segment, j , as shown in Figure 7.2. The two possibilities will be treated in the same way.

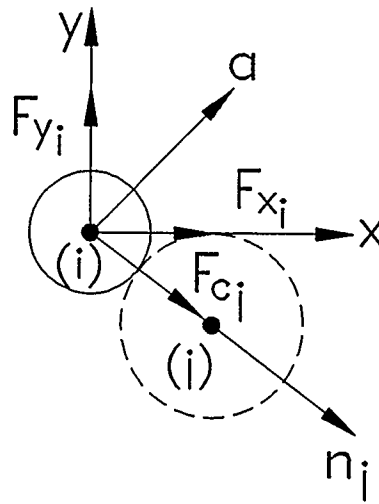


Figure 7.2. Diagram for determination of contact forces for a disk with one point of contact.

In the figure η_j -axis passes through the center of disk i and point j . This axis represents the direction of the contact force. We will define the following variables:

X_i, Y_i - coordinates of disk i .

X_j, Y_j - coordinates of point j , i.e. center of a disk, if the entity in contact is a disk, or the coordinates of the point of contact, if the entity is an obstruction line segment.

l_j, m_j - directional cosines for "j" direction, i.e. cosines of the angles between η_j -axis and the X- and Y-axis respectively.

L_{ji} - a characteristic length

$L_{ji} = R_i + R_j$ - if j is a disk

$L_{ji} = R_i$ - if j is an obstruction line segment

M_i - mass of disk i.

We define the directional cosines as:

$$l_j = \frac{X_j - X_i}{L_{ji}}, \quad m_j = \frac{Y_j - Y_i}{L_{ji}}$$

and we can write the dynamic equilibrium equation in η_j direction as

$$F_{cj} + l_j \cdot F_{xi} + m_j \cdot F_{yi} = M_i (l_j \cdot a_{xi} + m_j \cdot a_{yi})$$

Hence the contact force is determined as:

$$F_{cj} = l_j \cdot (M_i a_{xi} - F_{xi}) + m_j \cdot (M_i a_{yi} - F_{yi}) \quad (7.1)$$

7.2.2. Disks with Two Points of Contact

Let us assume that a given disk i is in contact with two entities j and k , as shown in Figure 7.2.

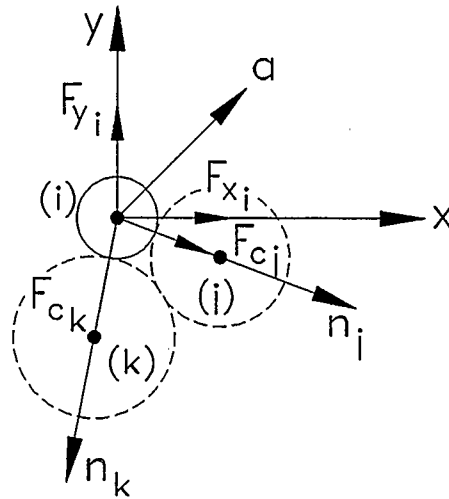


Figure 7.3. Diagram for determination of contact forces for a disk with two points of contact

The two contact forces are denoted by F_{cj} , F_{ck} , and their corresponding directions as η_j , η_k . Using the same notation as in Section 7.2.1 we can write the equilibrium equations in the two directions, similarly to Equation (7.1), as

$$F_{cj} + l_{jk} \cdot F_{ck} = l_j \cdot (M_i a_{xi} - F_{xi}) + m_j \cdot (M_i a_{yi} - F_{yi})$$

and

$$F_{ck} + l_{jk} \cdot F_{cj} = l_k \cdot (M_i a_{xi} - F_{xi}) + m_k \cdot (M_i a_{yi} - F_{yi})$$

where l_{jk} is the cosine of the angle between the two directions, i.e.

$$l_{jk} = \cos \angle (\eta_j, \eta_k) = \frac{(X_j - X_i)(X_k - X_i) + (Y_j - Y_i)(Y_k - Y_i)}{L_{ij} \cdot L_{ik}}$$

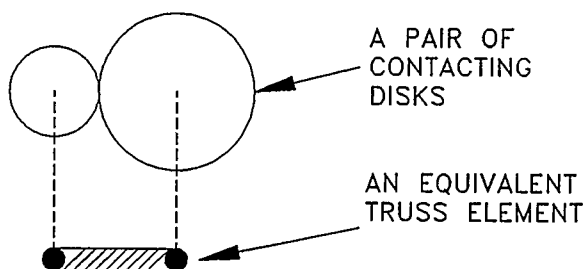
The equilibrium equations in two unknown contact forces become linearly dependent for l_{jk} equal to 1 or -1, in which case they cannot be solved. The condition that no overlap between the disks can occur, suggests that the directional cosine cannot be equal to 1. However, l_{jk} can be equal to -1. In this case the two directions (namely, η_j, η_k) coincide and effectively we only have one equation. Hence, the contact forces cannot be determined. If however, $l_{jk} \neq -1$, then the equilibrium equations can be solved. The contact forces can then be evaluated according to the following relations:

$$F_{cj} = \frac{1}{1 - l_{jk}^2} \left[(M_i a_{xi} - F_{xi})(l_j - l_k l_{jk}) + (M_i a_{yi} - F_{yi})(m_j - m_k l_{jk}) \right] \quad (7.2)$$

$$F_{ck} = \frac{1}{1 - l_{jk}^2} \left[(M_i a_{xi} - F_{xi})(l_k - l_j l_{jk}) + (M_i a_{yi} - F_{yi})(m_k - m_j l_{jk}) \right]$$

7.3 STATIC EQUILIBRIUM ANALYSIS

The following method of evaluating the contact forces among the disks present in the system is based on converting the conglomeration of the disks into an equivalent truss system. In the system, the center of each disk represents a node to which all the external forces acting on the given disk (F_x, F_y) , as well as the disk's inertial forces $(-ma_x, -ma_y)$, are applied. Each pair of contacting disks is replaced by a truss element spanning between the respective nodes, as shown below.



The length of a truss element is equal to the sum of the radii of the two disks that the truss replaces, i.e.: $l_{ij} = R_i + R_j$. The spatial positions of all the nodes are, thus, the same as those for the original disks. The base disks, i.e. those which are in contact with obstruction line segments are replaced by shorter truss elements running between the centers of those disks and the respective points of contact. Consequently, the points of contact between the

disks and the obstruction line segments become the additional nodes. A sample configuration of disks and the corresponding truss system are shown in Figure 7.4. The unknown contact forces are equivalent to the forces in the corresponding trusses.

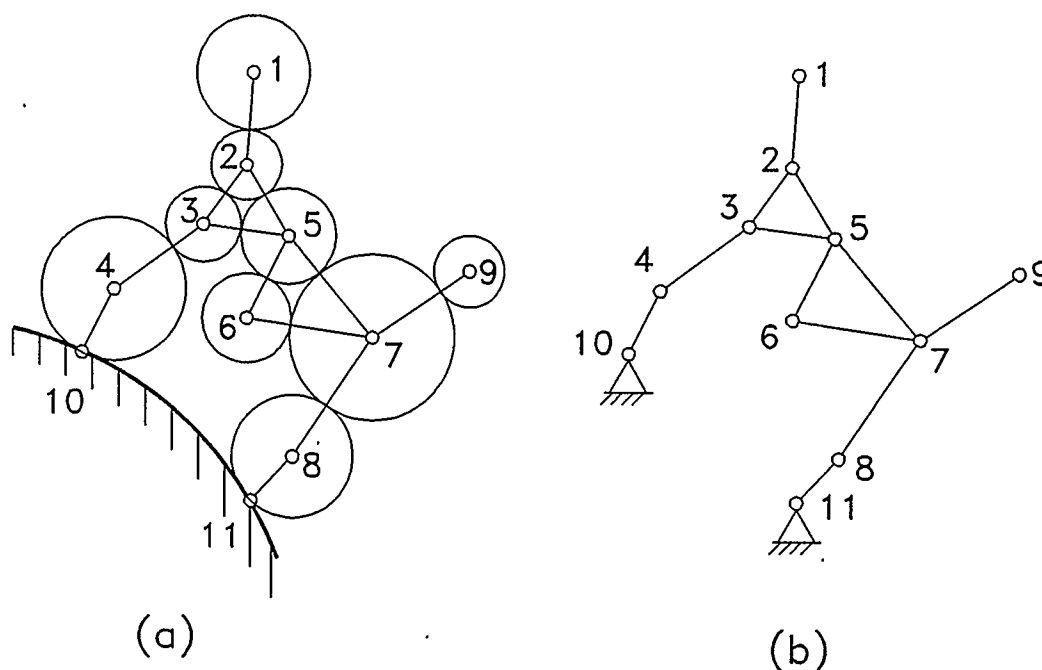


Figure 7.4. An example of a disk conglomeration (a), and the corresponding equivalent truss system (b).

The advantage of setting up the truss system in the manner described above, is that the truss system can be mapped directly from the topology of the disk conglomerate, and without any changes to its geometry.

The analysis of the equivalent truss system which

replaces the given configuration of disks is carried out by what is known as the matrix structure analysis or a direct method of finite element analysis [3]. This method is based on solving a system of algebraic equations for the nodal displacements of the structure after the load is applied to it. The force in each truss element is determined from the element's change in length resulting from the relative displacements of the two nodes that span it. Although this method can be applied to a variety of different structures, including frames and plates, we shall limit ourselves only to analyzing two-dimensional truss systems.

In this section an introductory theory with its major assumptions is briefly presented. The method of setting up the system of algebraic equations in terms of displacements, and its subsequent solution, is then discussed.

7.3.1 Introductory Theory

A matrix structure analysis is based on the assumption that the material composing the structure is in the linear (elastic) region at any time during the process, i.e. that no plastic deformations occur. From the linearity assumption it follows that the load-deflection relationship for the structure is linear. In other words, if all the external forces acting on the structure are multiplied by some constant C , then all the deflections of the structure will be " C " times the previous deflections. This also means that the total deflection of a structure at some point from all the external forces, will be equal to the sum of the deflections from the individual forces. It is also assumed that the forces are applied to the structure in a quasi-linear manner, that is the application is gradual, starting from zero and steadily increasing until the final value is reached.

Moreover, it is assumed that the deflections that occur are small enough not to cause any significant changes in geometry, i.e. that the geometry of the structure can be considered constant at all times during the deflection.

Let us consider an elastic linear body to which the forces are applied in a quasi-linear manner, as shown in

Figure 7.5. Let P_i represent the external forces applied to the body, and Δ_i be the corresponding displacements at the points at which the forces are applied and in the direction of the forces.

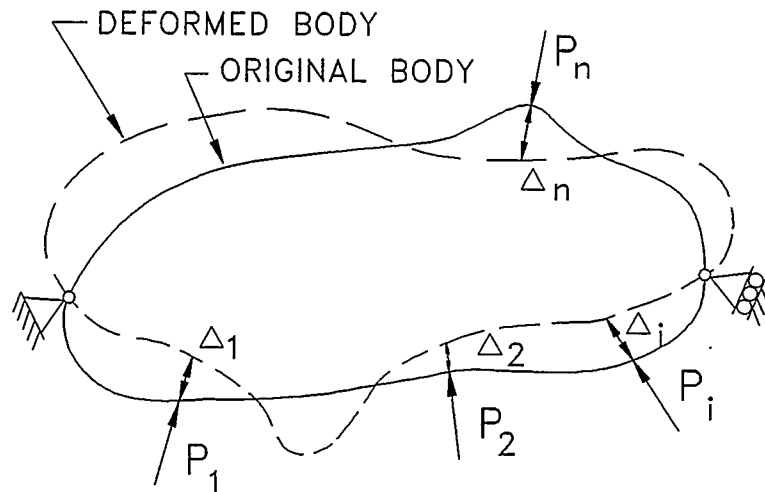


Figure 7.5. A schematic diagram for the formulation of the matrix equations

If the linearity principle holds, then the total work done by the external forces on the body, using the summation convention for repeated indices, can be written as:

$$U = \frac{1}{2} P_i \cdot \Delta_i \quad (7.3)$$

From the conservation energy the work done on the body by the external forces is equal to the strain energy of the body. If, now, a small variation of the strain is introduced to, say, Δ_i , then the corresponding variation of the strain energy

can be correlated according to:

$$\frac{\partial U}{\partial \Delta_i} = \frac{1}{2} P_i + \frac{1}{2} \Delta_j \cdot \frac{\partial P_j}{\partial \Delta_i} \quad (7.4)$$

According to Castiglano's first theorem, however,

$$\frac{\partial U}{\partial \Delta_i} = P_i \quad (7.5)$$

and therefore Equation (7.4) may be written as:

$$P_i = \Delta_j \cdot \frac{\partial P_j}{\partial \Delta_i}, \quad (i, j = 1, N) \quad (7.6)$$

Equation (7.6) represents a system of N simultaneous algebraic equations in N unknown displacements Δ_i . If we designate

$$K_{ij} = \frac{\partial P_j}{\partial \Delta_i} \quad (7.7)$$

then Equation (7.6) can be conveniently written in a matrix form as,

$$[P] = [K] \cdot [\Delta] \quad (7.8)$$

The term on the LHS of Equation (7.8) is known since it represents the known vector of external forces applied to the body.

Physically, the coefficients of the matrix on the RHS of the equation (K_{ij}) as defined by Equation (7.7) represent the holding force required at point j in order to keep the body in equilibrium, after a unit displacement is introduced at point i . Figure 7.6 illustrates this point.

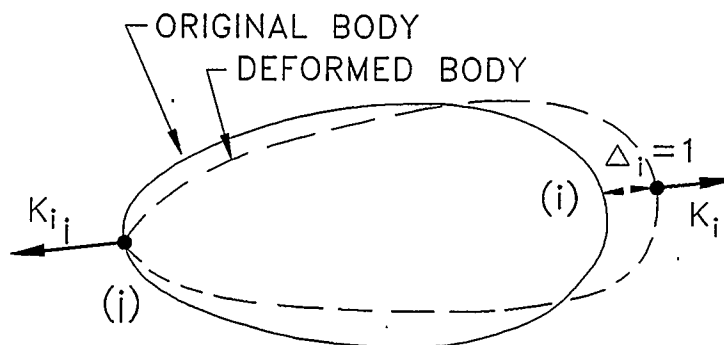


Figure 7.6. Physical interpretation of the coefficients of matrix K .

It, therefore, becomes clear that the coefficients K_{ij} depend only on the geometry of the body, the points at which the external loads are applied, and the elastic properties of the particular material which makes up the body. Since these are known, coefficients K_{ij} can always be computed.

Matrix $[K]$ is usually referred to as the global stiffness matrix of a body or a structure. After the assembly of the stiffness matrix the system of the algebraic equations (7.8) can be solved for the displacements.

A global stiffness matrix is a special type of matrix. Visualising its properties can lead to the possibility of using the numerical procedures which will require far less computational time.

It can be observed that the global stiffness matrix is symmetric, i.e. $K_{ij} = K_{ji}$. Namely, by Equation (7.5) we have

$$K_{ij} = \frac{\partial P_j}{\partial \Delta_i} = \frac{\partial}{\partial \Delta_i} \left(\frac{\partial U}{\partial \Delta_j} \right) = \frac{\partial^2 U}{\partial \Delta_i \partial \Delta_j} = \frac{\partial}{\partial \Delta_j} \left(\frac{\partial U}{\partial \Delta_i} \right) = \frac{\partial P_i}{\partial \Delta_j} = K_{ji}$$

Most of the effort during the matrix structural analysis goes into the assembly of the structure stiffness matrix. In terms of computational time involved, the symmetry of the matrix enables a considerable reduction, as only half of the matrix elements need to be evaluated.

The global stiffness matrix is positive definite. A matrix $[A]$ is positive definite if and only if, for any real vector $[x] \neq [0]$ the following condition is satisfied

$$[x]^T [A] [x] > 0 \quad (7.9)$$

The above condition, using a summation convention for repeated indices, can be written for the stiffness matrix as

$$x_i \cdot K_{ij} \cdot x_j > 0 \quad (7.10)$$

If x_i are thought of as the displacements of some body subjected to external forces P_i , then by Equation (7.7), Equation (7.6), and Equation (7.3)

$$x_i \cdot K_{ij} \cdot x_j = x_i \cdot P_i = 2U \quad (7.11)$$

And, evidently condition (7.10) is satisfied, since the strain energy is always greater than zero for any non-singular set of real displacements.

When analyzing a structure, using a stiffness matrix method, one typically divides the structure into elements. Those could be bars, trusses, frames, plates, blocks, etc. One also identifies the key points within the structure which are referred to as nodes. The nodes are located on the interfaces or the boundaries between the individual elements, and are the points at which the structure's deflections are to be computed. The proper choice of nodes and elements is largely a matter of experience, and, in itself, constitutes an art in using the stiffness matrix method for structural analysis.

It is also assumed that the external loads can only be applied to the nodes, that is, a force cannot be applied in the "middle" of an element. Consequently, the system of equations for nodal displacements, as given by Equation (7.8), can be set-up once the appropriate stiffness matrix is

evaluated. The arrays appearing in Equation (7.8) are referred to as:

- [P] - generalized force vector
- [K] - nodal structure stiffness matrix
- [Δ] - generalized nodal displacement vector.

The stiffness matrix is made up of the stiffness of the individual elements that comprise the structure. An entry of this matrix K_{ij} represents a generalized holding force required at node i after a unit generalized displacement is introduced at node j . In other words, K_{ij} represents a stiffness of the structural element spanned between nodes i and j . An entry on the main diagonal of the matrix K_{ii} represents a generalized force at node i required to cause a unit generalized displacement at this node.

There are two basic conditions that have to be satisfied at each node. Those are the displacement compatibility condition and the force equilibrium. The displacement compatibility states that the displacements at a given node for all structural elements joining at this node are the same and equal to the total displacement of the node. This condition can be written as:

$$\Delta_i^j = \Delta_i^k = \Delta_i^m = \cdots = \Delta_i^n = \Delta_i \quad (7.12)$$

where Δ_i^j denotes a generalized displacement of node i of element $i-j$. This statement is equivalent to the condition that the structure at a given node remains continuous and the elements involved do not separate from one another.

The force equilibrium states that the external force applied to a node is balanced by the internal forces of the elements joining at the node. Using the same notation as for the displacement compatibility, this condition can be written as:

$$P_i = P_i^j + P_i^k + P_i^m + \cdots + P_i^n \quad (7.13)$$

Let us now consider a single element $i-j$, as shown on Figure (7.7) below.

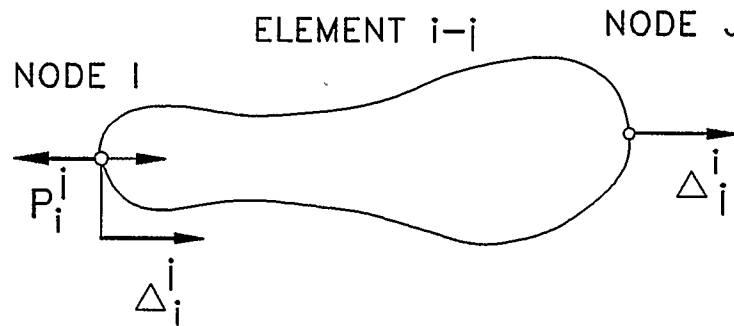


Figure 7.7. A diagram for evaluating a force in a single element.

It is evident that since the material is linear, the contributions from the displacements at the two nodes can be

added. Hence, using the definition of the coefficients of the stiffness matrix, the element force at node i is given by:

$$P_i^j = K_{ii}^j \cdot \Delta_i^j + K_{ij} \cdot \Delta_j^i \quad (7.14)$$

Substituting the above result into force equilibrium equation (7.13) and making use of displacement compatibility equation (7.12), we may write

$$P_i = \Delta_i \cdot \sum_x (K_{ii}^x) + \sum_x (K_{ix} \cdot \Delta_x) \quad (7.15)$$

where summation over x indicates a summation over all structure elements joined at node i , i.e. $x = j, k, m, \dots, n$. By comparing the result of Equation (7.15) with Equation (7.8) it becomes evident that the coefficients of the stiffness matrix on the main diagonal K_{ii} are the sums of the corresponding stiffnesses K_{ii}^x of all contributing elements $i-x$. We may therefore write

$$K_{ii} = K_{ii}^j + K_{ii}^k + K_{ii}^m + \dots + K_{ii}^n \quad (7.16)$$

Typically, coefficients K_{ii} are referred to as the nodal stiffnesses, while coefficients K_{ij} are termed as element stiffnesses.

The global stiffness matrix can now be assembled from the stiffnesses of the individual elements in the following way:

Step 1. Number all the nodes in the structure from 1 to N , where N represents the total number of nodes.

Step 2. For every i between 1 and N repeat step 3.

Step 3. For every j between $i+1$ and N do the following if nodes i and j span an element:

- calculate K_{ii}^j and add it to the i,i location of the global stiffness matrix $[K]$.
- calculate K_{jj}^i and add it to the j,j location of $[K]$.
- calculate K_{ij} and place it in the i,j location of $[K]$.
- set $K_{ji} = K_{ij}$ (symmetry).

The problem of setting up a system of equations for the nodal displacements in the structure was thus reduced to evaluating the appropriate nodal and element stiffnesses for each identified structural element.

For three dimensional structures the generalized nodal displacements and the generalized forces can have up to six components - three translational and three rotational. Consequently, the element and nodal stiffnesses K_{ij} , themselves, represent matrices with up to six rows and six columns. We will refer to these as the elemental stiffness matrices. The fact, that the global stiffness matrix is composed of the elemental stiffness matrices rather than single numbers, does not reduce the generality of the above

considerations since our problem is a linear one. For a general three-dimensional case we, simply, have $6N$ equations in $6N$ real displacements.

It may often be convenient to evaluate the elemental stiffnesses in a local coordinate system. These can then be transferred into the global system of coordinates. To demonstrate this, let us consider a system of equations expressed in local coordinates.

$$[p] = [k] \cdot [\delta] \quad (7.17)$$

The above equation should also be valid in the global system of coordinates. We write:

$$[P] = [K] \cdot [\Delta] \quad (7.18)$$

If matrix $[R]$ represents a transformation from local to global coordinates then $[p] = [R] \cdot [P]$, $[\delta] = [R] \cdot [\Delta]$, and we may write Equation (7.17) as

$$[R] \cdot [P] = [k] \cdot [R] \cdot [\Delta]$$

Substituting for $[P]$ from Equation (7.18) and rearranging the terms,

$$([R] \cdot [K] - [k] \cdot [R]) \cdot [\Delta] = 0$$

Since the above equation is satisfied for any arbitrary real vector $[\Delta]$, it follows that the term in brackets must vanish. Therefore,

$$[K] = [R^{-1}] \cdot [k] \cdot [R] \quad (7.19)$$

For a cartesian system of coordinates the inverse of the transformation matrix is equal to the transpose of the matrix, ($[R^{-1}] = [R^T]$). Using the result of Equation (7.19), we may finally write an expression for the elemental stiffness matrices.

$$\left. \begin{aligned} [K_{ii}^j] &= [R_{ij}^T] \cdot [k_{ii}^j] \cdot [R_{ij}] \\ [K_{ij}] &= [R_{ij}^T] \cdot [k_{ij}] \cdot [R_{ji}] \end{aligned} \right\} \quad (7.20)$$

In the above expression $[R_{ij}]$ represents a transformation matrix from local coordinate system chosen at node i for element $i-j$, into the global system of coordinates. It should be pointed out that the choice of coordinate system maybe different at the two nodes, hence, $[R_{ji}]$ in Equation (7.20).

In our model, we are dealing only with equivalent two-dimensional truss systems. The discussion of method of evaluation of the elemental stiffness matrices for general three-dimensional elements is therefore beyond the scope of this section. The implementation of the general methodology described here, to the equivalent truss systems will be

presented in the following sub-sections.

As a final note, we will briefly discuss the aspect of boundary conditions that may be prescribed on the structure. Typically, the boundary conditions are specified in terms of certain required nodal displacements (nodal constraints). For example, fixed supports at certain nodes, would necessitate that the nodal displacements at these points be zero. If a boundary condition prescribes a zero displacement at some node, say $\Delta_i = 0$, then clearly, this displacement does not contribute to any of the external forces in Equation (7.8) (the contribution is represented here by the i -th column of the stiffness matrix $[K]$). Moreover, the equation corresponding to this node (given by the i -th row of matrix $[K]$) becomes redundant. Consequently, the boundary condition is implemented by deleting the i -th row and the i -th column from the global stiffness matrix. The new reduced system of equations will then represent the constrained structure.

7.3.2 Equivalent Truss System

As mentioned before, a conglomeration of disks for which the contact forces cannot be computed using the dynamic equilibrium, is transformed into the equivalent two-dimensional truss system. Such a system is created by placing a node at the center of each disk, and by replacing each pair of contacting disks with a truss element which spans the two nodes. The length of a truss element is thus, equal to the sum of the radii of the two disks that the given element replaced.

Each point of contact between a disk and a boundary line segment is also made into a node. We will refer to such a node as a base node. The contacting disk is replaced by a truss which runs between the node at the center of the disk and the given base node. Such truss element will be called a base element. The length of a base element is equal to the radius of the disk that it replaces. We assume that a base node is constrained, that is that zero displacements are prescribed at this node. Therefore the base nodes will not contribute any equations to the global system, see Section 7.3.1. The base elements, however, will contribute to the nodal stiffness of its other (non-base) node. Figure (7.8) shows the definition of the elements described above.

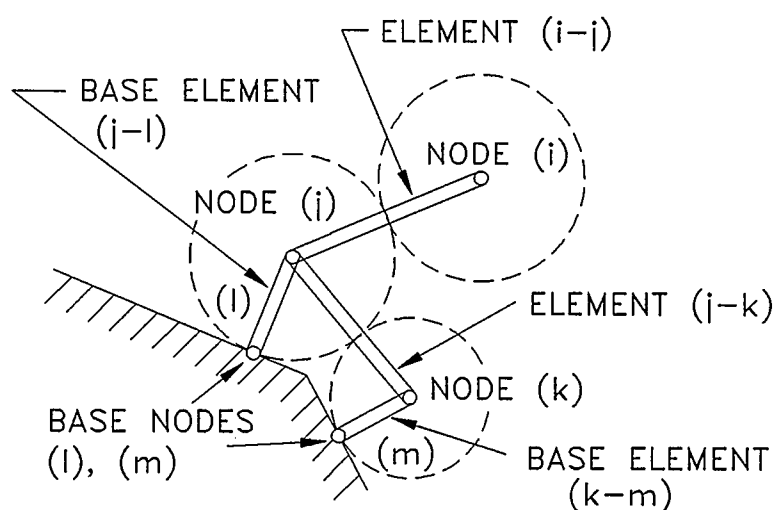


Figure 7.8. A definition of the elements in the equivalent truss system.

The non-base nodes are numbered from 1 to N while the base nodes are numbered from $N+1$ to $N+N_B$. The global coordinates of the nodes are the same as the coordinates of the centers of respective disks, which are always known. We will denote them as X_i, Y_i .

The length of each truss element is also known. The length of truss element $i-j$ is defined as: $L_{ij} = R_i + R_j$.

For two-dimensional truss systems the generalized nodal displacements Δ_i can have only two translational components. The situation is similar for the generalized forces P_i . Hence, the element and nodal stiffnesses K_{ij}, K_{ii} , are represented by two-dimensional square matrices. We write,

$$\Delta_i = \begin{bmatrix} \Delta_x \\ \Delta_y \end{bmatrix}_i, \quad P_i = \begin{bmatrix} P_x \\ P_y \end{bmatrix}_i, \quad K_{ij} = \begin{bmatrix} K_{11} & K_{12} \\ K_{21} & K_{22} \end{bmatrix}_{ij} \quad (7.21)$$

It is evident that the condition for the equilibrium of forces at each node will consist of two corresponding equations (one for each of the two force components). Consequently, there will be $2N$ equations in $2N$ unknown real displacements. The system of equations is organized in such a way that the equilibrium equations for a node will occupy two consecutive lines. Therein, the equilibrium equations for node 1 will occupy lines 1 and 2, while those for, say, node i will be located on lines $2i-1$ and $2i$. Therefore, the following allocation of variables will be used:

$$\begin{bmatrix} \Delta_x \\ \Delta_y \end{bmatrix}_i \rightarrow \begin{bmatrix} \Delta_{2i-1} \\ \Delta_{2i} \end{bmatrix}, \quad \begin{bmatrix} P_x \\ P_y \end{bmatrix}_i \rightarrow \begin{bmatrix} P_{2i-1} \\ P_{2i} \end{bmatrix} \quad (7.22)$$

and

$$\begin{bmatrix} K_{11} & K_{12} \\ K_{21} & K_{22} \end{bmatrix}_{ij} \rightarrow \begin{bmatrix} K_{2i-1, 2j-1} & K_{2i-1, 2j} \\ K_{2i, 2j-1} & K_{2i, 2j} \end{bmatrix} \quad (7.23)$$

The elemental stiffness matrices of Equation (7.23) can be obtained by first calculating them in a conveniently chosen local system of coordinates and then transforming them into the global system by the application of Equation (7.20) derived in Section 7.3.1.

For a truss element we chose a local coordinate system at each node in the following manner. The origin of the system is located at the given node. The x -axis runs along the length of the element and in the direction away from the node. Figure (7.9) shows our choice of local coordinates at the respective nodes.

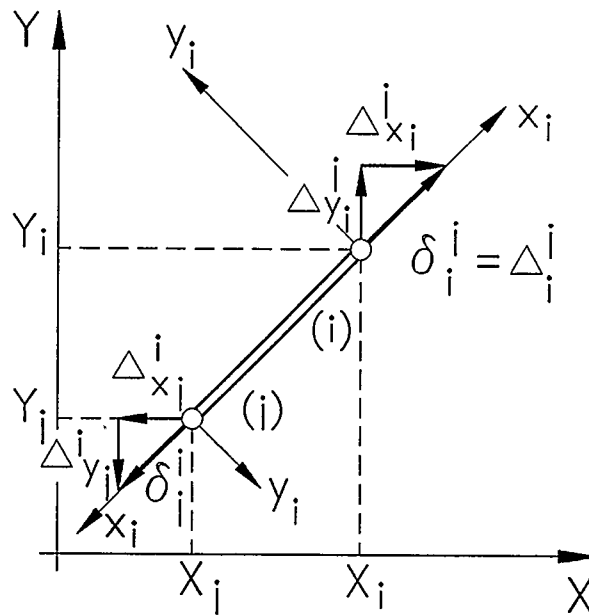


Figure 7.9. Definition of local systems of coordinates and the corresponding displacements.

The choice of the local coordinate systems at the two nodes with different orientation of axis, as shown in Figure 7.9, enables us to treat each node in the same way.

The coordinate transformation matrix describing a rotation of the cartesian system of coordinates $[R_{ij}]$ is given

by:

$$[R_{ij}] = \begin{bmatrix} l_1 & m_1 \\ l_2 & m_2 \end{bmatrix}_{ij} \quad (7.24)$$

where l_1, m_1 are the directional cosines of x-axis with X- and Y-axis, and l_2, m_2 the directional cosines of y-axis with X- and Y-axis, respectively. Subscripts ij indicate a local system of coordinates chosen for node i of element $i-j$.

From the definition of the coordinate systems it becomes evident that

$$[R_{ji}] = -[R_{ij}] \quad (7.25)$$

Since a truss element can experience only a change of length, in the local system of coordinates the nodal displacement δ_i^j takes place only in x direction. That is, only x component of the displacement is present. Similarly, the forces induced at the nodes by the nodal displacements, can only be directed along the truss, and thus in local coordinates, have only the x component present. Consequently, the local elemental stiffness matrix $[k_{ij}]$ will have only one non-zero entry, located in the first row and the first column. This is demonstrated by the following relation

$$\begin{bmatrix} p_x \\ 0 \end{bmatrix}_{ij} = \begin{bmatrix} k_{11} & 0 \\ 0 & 0 \end{bmatrix}_{ij} \cdot \begin{bmatrix} \delta_x \\ 0 \end{bmatrix}_j$$

We can, therefore, drop the second row in the above equation (all zeros), as well as the second column of the elemental stiffness matrix. Hence, we will write

$$p_{ij} = k_{ij} \cdot \delta_j^i \quad (7.25)$$

p_{ij} denotes here, a force induced at node i of element $i-j$ by a displacement δ_j^i at node j , while maintaining the equilibrium of the element. Using this convention we can also write:

$$p_{ii}^j = k_{ii}^j \cdot \delta_i^j \quad (7.26)$$

The above equations, (7.25) and (7.26), represent a simple load-deflection relation for a linear member. Consequently, for a uniform cross-section truss element we have a very familiar expression for the stiffnesses,

$$k_{ii}^j = k_{ij} = \left(\frac{EA}{L} \right)_{ij} \quad (7.27)$$

where E is the elastic modulus of the material, A is the cross-sectional area of the truss element, and L is its length.

If we recall that if $[R]$ represents the coordinate transformation matrix from a global to a local systems then, $[\delta] = [R] \cdot [\Delta]$, $[p] = [R] \cdot [P]$. For the truss element we can

write this as

$$\begin{bmatrix} \delta_x \\ 0 \end{bmatrix}_{ij} = \begin{bmatrix} l_1 & m_1 \\ l_2 & m_2 \end{bmatrix}_{ij} \cdot \begin{bmatrix} \Delta_x \\ \Delta_y \end{bmatrix}_i, \quad \begin{bmatrix} P_x \\ 0 \end{bmatrix}_{ij} = \begin{bmatrix} l_1 & m_1 \\ l_2 & m_2 \end{bmatrix}_{ij} \cdot \begin{bmatrix} P_x \\ P_y \end{bmatrix}_i$$

It is evident that the second equation in the above relation is redundant ($0 = 0$), and thus can be deleted. Hence, in our case, the coordinate transformation matrix is given only by the top row, i.e. $[l_1 \ m_1]$. We may therefore write

$$[R_{ij}] = [l \ m]_{ij}, \quad [R_{ij}^T] = \begin{bmatrix} l \\ m \end{bmatrix}_{ij}, \quad [R_{ji}] = [-l \ -m]_{ij} \quad (7.28)$$

The directional cosines of Equation (7.28) are evaluated from the spacial position of the truss in the global coordinate system. Namely,

$$l = \frac{X_i - X_j}{L_{ij}}, \quad m = \frac{Y_i - Y_j}{L_{ij}} \quad (7.29)$$

Now, combining Equation (7.20), Equation (7.27) and Equation (7.28) we can obtain the expressions for the elemental stiffness matrices in global coordinates. We have,

$$[K_{ij}] = \begin{bmatrix} l \\ m \end{bmatrix}_{ij} \cdot \left(\frac{EA}{L} \right)_{ij} \cdot [-l \ -m]_{ij}$$

and

$$[K_{ii}^j] = \begin{bmatrix} 1 \\ m \end{bmatrix}_{ij} \cdot \left(\frac{EA}{L} \right)_{ij} \cdot [1 \ m]_{ij}$$

We define a two-dimensional square matrix $[B_{ij}]$ as:

$$[B_{ij}] = \begin{bmatrix} 1^2 & 1m \\ 1m & m^2 \end{bmatrix}_{ij} \quad (7.30)$$

Using the definition of Equation (7.30) we can write the expressions for elemental stiffnesses as

$$[K_{ij}] = - \left(\frac{EA}{L} \right)_{ij} \cdot [B_{ij}] \quad (7.31)$$

and

$$[K_{ii}^j] = \left(\frac{EA}{L} \right)_{ij} \cdot [B_{ij}] \quad (7.32)$$

With our choice of the local system of coordinates, node j of truss element i - j is treated in the same way as node i . We could, therefore, by Equation (7.20), write for node j :

$$[K_{jj}^i] = \begin{bmatrix} -1 \\ -m \end{bmatrix}_{ij} \cdot \left(\frac{EA}{L} \right)_{ij} \cdot [-1 \ -m]_{ij}$$

and in short,

$$[K_{jj}^i] = \left(\frac{EA}{L} \right)_{ij} \cdot [B_{ij}] \quad (7.33)$$

Equations (7.31), (7.32), and (7.33) define the elemental stiffness matrices. The global stiffness matrix can now be

assembled from the elemental matrices, provided that the equivalent stiffness of the truss element $(EA/L)_{ij}$ is known. The computation of the equivalent stiffnesses will be discussed in Section 7.3.3 and the method of assembly of the global stiffness matrix will be presented in Section 7.3.4.

Finally, the system of equations for the nodal displacements can be set up once the vector representing the external forces $[P]$ is determined. A conglomeration of disks is a dynamic system, with component disks experiencing accelerations. On the other hand, the equivalent truss system, for the application with the present method, must be quasi-static. Hence, the dynamic terms of acceleration present in a disk conglomerate must be, therefore, converted to inertia forces. This is done by adding the dynamic term (mass times acceleration) of a disk to the external force acting on it. Henceforth, for node i , which replaces disk i , we have:

$$[P_i] = \begin{bmatrix} P_x \\ P_y \end{bmatrix}_i = \begin{bmatrix} F_x - ma_x \\ F_y - ma_y \end{bmatrix}_i \quad (7.34)$$

where: F_x, F_y are the components of the total external force acting on disk i , a_x, a_y are the components of the acceleration experienced by the center of the disk, and m represents the disk's mass.

The system of linear algebraic equations for the nodal displacements of the equivalent truss system will be of the following form:

$$\begin{bmatrix} P_1 \\ P_2 \\ \vdots \\ P_{2i-1} \\ P_{2i} \\ \vdots \\ P_{2N-1} \\ P_{2N} \end{bmatrix} = \begin{bmatrix} K_{11} & K_{12} & \cdot & K_{12j-1} & K_{12j} & \cdot \cdot & K_{12N-1} & K_{12N} \\ K_{21} & K_{22} & \cdot & K_{22j-1} & K_{22j} & \cdot \cdot & K_{22N-1} & K_{22N} \\ \cdot & \cdot & \cdot & \cdot & \cdot & \cdot \cdot & \cdot & \cdot \\ K_{2i-11} & K_{2i-12} & \cdot & K_{2i-12j-1} & K_{2i-12j} & \cdot \cdot & K_{2i-12N-1} & K_{2i-12N} \\ K_{2i1} & K_{2i2} & \cdot & K_{2i2j-1} & K_{2i2j} & \cdot \cdot & K_{2i2N-1} & K_{2i2N} \\ \cdot & \cdot & \cdot & \cdot & \cdot & \cdot \cdot & \cdot & \cdot \\ \cdot & \cdot & \cdot & \cdot & \cdot & \cdot \cdot & \cdot & \cdot \\ K_{2N-11} & K_{2N-12} & \cdot & K_{2N-12j-1} & K_{2N-12j} & \cdot \cdot & K_{2N-12N-1} & K_{2N-12N} \\ K_{2N1} & K_{2N2} & \cdot & K_{2N2j-1} & K_{2N2j} & \cdot \cdot & K_{2N2N-1} & K_{2N2N} \end{bmatrix} \cdot \begin{bmatrix} \Delta_1 \\ \Delta_2 \\ \vdots \\ \Delta_{2j-1} \\ \Delta_{2j} \\ \vdots \\ \Delta_{2N-1} \\ \Delta_{2N} \end{bmatrix} \quad (7.35)$$

The coefficients of the global stiffness matrix in Equation (7.35) are explained by the allocation condition (7.23). For example,

$$\begin{bmatrix} K_{2N-1, 2j-1} & K_{2N-1, 2j} \\ K_{2N, 2j-1} & K_{2N, 2j} \end{bmatrix} = \begin{bmatrix} K_{11} & K_{12} \\ K_{21} & K_{22} \end{bmatrix}_{Nj} = [K_{Nj}]$$

represents the element stiffness matrix for truss element $N-j$, as given by Equation (7.31).

The force and the displacement vectors in Equation (7.35) are given by the variable allocation conditions (7.22), i.e.

$$\begin{bmatrix} P_1 \\ P_2 \\ \cdot \\ P_{2i-1} \\ P_{2i} \\ \cdot \\ \cdot \\ P_{2N-1} \\ P_{2N} \end{bmatrix} = \begin{bmatrix} P_{x1} \\ P_{y1} \\ \cdot \\ P_{xi} \\ P_{yi} \\ \cdot \\ \cdot \\ P_{xN} \\ P_{yN} \end{bmatrix}, \quad \begin{bmatrix} \Delta_{x1} \\ \Delta_{y1} \\ \cdot \\ \Delta_{xj} \\ \Delta_{yj} \\ \cdot \\ \cdot \\ \Delta_{xN} \\ \Delta_{yN} \end{bmatrix} = \begin{bmatrix} \Delta_1 \\ \Delta_2 \\ \cdot \\ \Delta_{2j-1} \\ \Delta_{2j} \\ \cdot \\ \cdot \\ \Delta_{2N-1} \\ \Delta_{2N} \end{bmatrix} \quad (7.36)$$

This concludes the consideration of the equivalent truss system which replaces the given disk conglomerate. The subject covered in this section represents an adaptation of the general theory presented in Section 7.3.1 to the equivalent truss system. In the following sections the above theory will be implemented to create a computer program which will set up the system of equations (7.35) and solve them for displacements. Consequently, the internal forces in the truss elements, which themselves represent the required contact forces, will be computed.

7.3.3 Equivalent Element Stiffness

The equivalent stiffness of a truss element $i-j$ is denoted as $(EA/L)_{ij}$, and represents the amount of force required at the nodes to cause a unit change in length of the element. Consider the diagram of Figure 7.10 below.

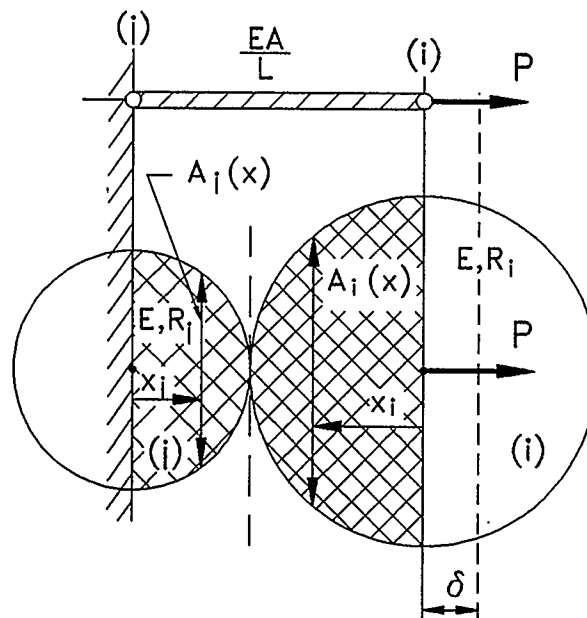


Figure 7.10. A diagram for evaluating the equivalent element stiffness.

For disk i we have,

$$\sigma_i(x) = \frac{P}{A_i(x)}, \quad A_i(x) = 2t_i \sqrt{R_i^2 - x_i^2}$$

Since we are dealing with a two-dimensional case, we will consider everything in terms of a unit thickness ($t_i = t_j = 1$). The strain for the disk is thus given by:

$$\epsilon_i(x) = \frac{\sigma_i(x)}{E} = \frac{P}{2E} \cdot \frac{1}{\sqrt{R_i^2 - x_i^2}}$$

Similarly for disk j ,

$$\epsilon_j(x) = \frac{P}{2E} \cdot \frac{1}{\sqrt{R_j^2 - x_j^2}}$$

Now, the change in length of the element is

$$\delta = \int_0^{R_i} \epsilon_i(x) dx_i + \int_0^{R_j} \epsilon_j(x) dx_j$$

and

$$\delta = \frac{P}{2E} \left[\int_0^{R_i} \frac{1}{\sqrt{R_i^2 - x_i^2}} dx_i + \int_0^{R_j} \frac{1}{\sqrt{R_j^2 - x_j^2}} dx_j \right] = \frac{\pi P}{2E}$$

Hence, the equivalent stiffness becomes,

$$\left(\frac{EA}{L} \right)_{ij} = \frac{2E}{\pi} \quad (7.37)$$

7.3.4 Assembly of the Global Stiffness Matrix

The equivalent truss system contains N non-base nodes and N_B base nodes. The non-base nodes are numbered from 1 to N , while the base nodes are numbered $N+1, N+2, \dots, N+N_B$. As pointed out earlier, the base nodes are assumed to be constrained. In other words, the generalized displacements at these nodes are zero. We can write this condition as

$$\Delta_{xj} = \Delta_{yj} = 0, \quad (N < j \leq N+N_B)$$

Therefore, as demonstrated in Section 7.3.1, the equilibrium equations for these nodes become redundant. Also, since the displacements corresponding to these nodes are zero, they do not contribute to the force equilibrium equations for any other nodes. Consequently, the elemental stiffness matrices, $[K_{ji}]$, $[K_{ij}]$, $[K_{jj}^i]$, need not be calculated. It follows from here that the global stiffness matrix $[K]$ will have the dimension of $2N$. It should, however, be pointed out that the existence of a base element connecting a non-base node i to a base node j implies that the nodal stiffness at i is affected. Hence, $[K_{ii}^j]$ has to be computed and included in the global stiffness matrix $[K]$.

In our program the topology of the equivalent truss system is given by the node connectivity array, $IC(i,j)$. The dimension of this array is $[N \times N+N_B]$. The square part of the array, $[N \times N]$, corresponds to the connectivities among the non-base nodes, while the rectangular part, $[N \times N_B]$, represents the connectivities between the non-base nodes and the base nodes. An element of this array can be either 1 or 0. $IC(i,j) = 1$ indicates that nodes i and j are connected by a truss element. Zero, of course, means that no such connection exists.

Using the procedure outlined in Section 7.3.1, the algorithm for the assembly of the global stiffness matrix of the equivalent truss system can now be summarized as follows.

Step 1. For $i = 1, 2, \dots, N-1$ do Steps 2-10

Step 2. For $j = i+1, i+2, \dots, N$ do Steps 3-9 provided that $IC(i, j) = 1$, i.e. an element exists between the two nodes (i, j) .

Step 3. Calculate the length of the element.

$$L_{ij} = R_i + R_j$$

Step 4. Calculate the equivalent element stiffness in local coordinates, Equation (7.37).

$$\left(\frac{EA}{L}\right)_{ij} = \frac{2E}{\pi}$$

Step 5. Calculate the coefficients of the coordinate transformation matrix, Equation (7.29).

$$l = \frac{X_i - X_j}{L_{ij}}, \quad m = \frac{Y_i - Y_j}{L_{ij}}$$

Step 6. Calculate the element stiffness $[K_{ij}]$ according to Equation (7.31) and place it in the appropriate location of $[K]$, Equation (7.23).

$$- \frac{2E}{\pi} \cdot \begin{bmatrix} l^2 & lm \\ lm & m^2 \end{bmatrix} \Rightarrow \begin{bmatrix} K_{2i-1 \ 2j-1} & K_{2i-1 \ 2j} \\ K_{2i \ 2j-1} & K_{2i \ 2j} \end{bmatrix}$$

Step 7. Set $[K_{ji}] = [K_{ij}]$ and place it in the global matrix $[K]$, Equation (7.23).

$$\begin{bmatrix} K_{2i-1 \ 2j-1} & K_{2i-1 \ 2j} \\ K_{2i \ 2j-1} & K_{2i \ 2j} \end{bmatrix} \Rightarrow \begin{bmatrix} K_{2j-1 \ 2i-1} & K_{2j-1 \ 2i} \\ K_{2j \ 2i-1} & K_{2j \ 2i} \end{bmatrix}$$

Step 8. Calculate the nodal stiffness $[K_{ii}^j]$ from Equation (7.32) and add it to the appropriate location of $[K]$.

$$\frac{2E}{\pi} \cdot \begin{bmatrix} l^2 & lm \\ lm & m^2 \end{bmatrix} + \begin{bmatrix} K_{2i-1 \ 2i-1} & K_{2i-1 \ 2i} \\ K_{2i \ 2i-1} & K_{2i \ 2i} \end{bmatrix} \Rightarrow \begin{bmatrix} K_{2i-1 \ 2i-1} & K_{2i-1 \ 2i} \\ K_{2i \ 2i-1} & K_{2i \ 2i} \end{bmatrix}$$

Step 9. Calculate the nodal stiffness $[K_{jj}^i]$ from Equation (7.33) and add it to $[K]$.

$$\frac{2E}{\pi} \cdot \begin{bmatrix} l^2 & lm \\ lm & m^2 \end{bmatrix} + \begin{bmatrix} K_{2j-1 \ 2j-1} & K_{2j-1 \ 2j} \\ K_{2j \ 2j-1} & K_{2j \ 2j} \end{bmatrix} \Rightarrow \begin{bmatrix} K_{2j-1 \ 2j-1} & K_{2j-1 \ 2j} \\ K_{2j \ 2j-1} & K_{2j \ 2j} \end{bmatrix}$$

Step 10. For $j = N+1, N+2, \dots, N+N_B$ do Steps 11 and 12 provided that $IC(i, j) = 1$.

Step 11. Calculate the length of the element.

$$L_{ij} = R_i$$

Step 12. Repeat Steps 4, 5, and 8.

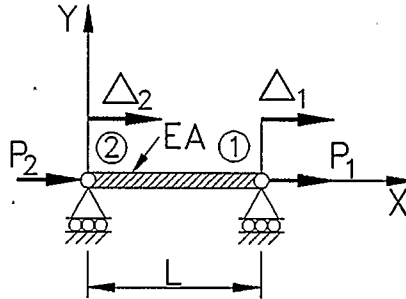
Step 13. For $i = N$ repeat Step 10.

The global stiffness matrix for the equivalent truss system is now assembled. The system of equations (7.35) can now be set up. The external force vector is determined from Equation (7.34) and allocated according to Equation (7.36). We will now proceed with describing the methodology of solving the equations for the nodal displacements.

7.3.5 Solution for Nodal Displacements

In this section a solution of the system of equations (7.35) will be discussed. The global stiffness matrix was assembled in Section 7.3.4. It was demonstrated in Section 7.3.1 that for the general case the stiffness matrix is positive definite. This statement holds true provided that the structure is in static equilibrium, i.e. that the number of degrees of freedom for the structure is zero. Clearly if this were not the case, then Equation (7.3), on which the entire static analysis is based, would, in addition to strain energy, also have had to include the change in kinetic and potential energies. Consequently, a stiffness matrix representing a system which is not in static equilibrium, would become singular. Since, regardless of the situation, a stiffness matrix is always symmetric, it follows that some rows of the matrix are proportional, or in general, that the rows are linearly dependent, as are the columns. This, in turn, implies that some of the equations for the nodal displacements are redundant and should be discarded. Discarding an equation for the nodal displacements is equivalent to eliminating the corresponding row and column from the stiffness matrix, which, in the physical sense, is analogous to putting a constraint on the corresponding displacement. This, of course is the same as adding an

appropriate support to the structure. This point can be explained by considering a simple example below.

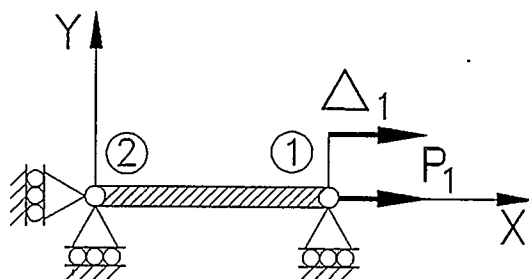


The structure shown in the sketch above, which consists of just one truss element, is not in static equilibrium. It has one degree of freedom, that being translation in the X direction. Because of the supports at nodes 1 and 2, the Y components of the displacements at the two nodes are zero. Hence, the system of equations for the nodal displacements of structure would have the following form:

$$\begin{bmatrix} P_1 \\ P_2 \end{bmatrix} = \begin{bmatrix} \frac{EA}{L} & -\frac{EA}{L} \\ -\frac{EA}{L} & \frac{EA}{L} \end{bmatrix} \cdot \begin{bmatrix} \Delta_1 \\ \Delta_2 \end{bmatrix}$$

The stiffness matrix in the above relation is clearly singular. Hence one of the above equations is redundant and should be eliminated. We may choose to discard the second one (i.e., the one for the force equilibrium at node 2), which we do by placing an appropriate support at node 2 (an appropriate constraint on a displacement at 2). This is shown in a sketch

below.



The new, "supported", structure is in static equilibrium, as it has zero degrees of freedom. We now have only one force equilibrium equation, that being for node 1 in the direction of X ,

$$P_1 = \frac{EA}{L} \Delta_1$$

The redundancy in force equilibrium equations in the example above was expected from the beginning. We had at our disposal an additional equation for the structure as a whole, i.e. one of the static equilibrium equations. Namely, that the sum of all external forces on the structure in X direction is equal to zero. This equation would provide us with condition $P_1 = P_2$. And hence, would reduce the system of two force equilibrium equations at the two nodes, to just one equation at one of the nodes. Effectively, adding the support to the structure was equivalent to utilizing the static equilibrium condition.

In general, for structures the number of degrees of freedom is equal to the number of additional supports (constraints on displacements) that are required so that the structure is in static equilibrium. Keeping the considerations of the previous paragraphs in mind, we could also state that the number of degrees of freedom of the structure is equal to the number of linearly dependent rows (or columns) in the stiffness matrix for this structure. Hence, the rank of the stiffness matrix is its dimension less the number of degrees of freedom. The converse also holds true. Namely, if the rank of the stiffness matrix was found to be less than the dimension of the matrix, by some number, then this number is equal to the number of degrees of freedom of the structure, which, in turn, represents the number of additional supports required. If we denote the number of degrees of freedom as $N_{D.F.}$, then for the equivalent truss system we can write:

$$N_{D.F.} = 2N - \text{rank}\{[K]\}$$

The equivalent truss system will very seldom be in static equilibrium. Consequently, the rank of the stiffness matrix will, almost always, be less than its dimension ($2N$), and hence some supports will have to be added. Once the rank of the global stiffness matrix is evaluated, then the appropriate number of constraints (given by the above condition) is added

to the system. The corresponding displacements are set to zero, and all the remaining displacements are solved for, from the reduced system of equations.

Even if the rank of the stiffness matrix is known beforehand, for the equivalent truss system in general, it is very difficult to determine at which nodes the constraints should be added to the system. The problem that presents itself here, is that after the addition of constraints and the subsequent deletion of the corresponding rows and columns from the global stiffness matrix, the resultant matrix must be composed of only linearly independent rows. Evidently, an addition of any arbitrary constraints will not guarantee the independence of the remaining equations. It is therefore proposed in this thesis that the determination of all the linearly independent equations (which is equivalent to the rank of the stiffness matrix) be attempted. Once this is accomplished all the remaining equations are disregarded and the corresponding displacements are set to zero.

Numerically, this can be carried out by applying a Gaussian elimination to the original system of equations to convert it to the one defined by an upper triangular matrix. Namely, the original system of equations which is given by

$$[P] = [K] \cdot [\Delta]$$

is reduced to

$$[P'] = [K'] \cdot [\Delta] \quad (7.38)$$

where $[K']$ is

$$[K'] = \begin{bmatrix} K'_{11} & K'_{12} & \cdot & \cdot & K'_{1r} & \cdot & K'_{12N} \\ 0 & K'_{22} & \cdot & \cdot & K'_{2r} & \cdot & K'_{22N} \\ 0 & 0 & K'_{33} & \cdot & K'_{3r} & \cdot & K'_{32N} \\ \cdot & \cdot & \cdot & \cdot & \cdot & \cdot & \cdot \\ \cdot & \cdot & \cdot & \cdot & \cdot & \cdot & \cdot \\ 0 & 0 & 0 & \cdot & K'_{rr} & \cdot & K'_{r2N} \\ 0 & 0 & 0 & \cdot & 0 & \cdot & 0 \\ \cdot & \cdot & \cdot & \cdot & \cdot & \cdot & \cdot \\ 0 & 0 & 0 & \cdot & 0 & \cdot & 0 \end{bmatrix} \quad (7.39)$$

The first say, r rows of the reduced matrix of Equation (7.39), which are not entirely composed of zeros, correspond to all the linearly independent equations. Hence, in the example above, the rank of the matrix is r . All the rows composed entirely of zeros represent the linearly dependent equations which are discarded. Consequently, the corresponding displacements $(\Delta_{r+1}, \Delta_{r+2}, \dots, \Delta_{2N})$ are set to zero. We have,

$$\Delta_{r+1} = \Delta_{r+2} = \dots = \Delta_{2N} = 0$$

The remaining displacements are obtained by solving the reduced system of equations:

$$\begin{bmatrix} P'_1 \\ P'_2 \\ \vdots \\ P'_{r-1} \\ P'_r \end{bmatrix} = \begin{bmatrix} K'_{11} & K'_{12} & \cdots & K'_{1\ r-1} & K'_{1\ r} \\ 0 & K'_{22} & \cdots & K'_{2\ r-1} & K'_{2\ r} \\ \vdots & \vdots & \ddots & \vdots & \vdots \\ 0 & 0 & \cdots & K'_{r-1\ r-1} & K'_{r-1\ r} \\ 0 & 0 & \cdots & 0 & K'_{r\ r} \end{bmatrix} \cdot \begin{bmatrix} \Delta_1 \\ \Delta_2 \\ \vdots \\ \Delta_{r-1} \\ \Delta_r \end{bmatrix}$$

The above system of equations can readily be solved through a backward substitution. Clearly, each equation in the above system has exactly one unknown less than the preceding equation, with the last equation having only one unknown. Hence, the last coefficient of the displacement vector, Δ_r , can be solved for. It is then substituted into the next, i.e. $r-1$ st, equation, which now has only one unknown, and the next displacement (Δ_{r-1}) is computed. The procedure is repeated by progressively stepping up to the higher equations and evaluating the corresponding displacements until the last of them, Δ_1 , is determined.

This concludes the determination of the displacements of the equivalent truss system. The forces in the truss elements, which are the same as the contact forces among the disks, can now be computed. This is discussed in the next section.

The algorithms for the Gaussian elimination and the backward substitution will not be discussed here. They are standard items in any introductory textbook on numerical methods, and therefore can easily be found (see [7], for example).

In closing it should be pointed out that during Gaussian elimination, one frequently refers to inter-changing the rows in the system of equations. This is done so that the elimination can still proceed, even though an element on the main diagonal of the matrix becomes zero. In the end, in the reduced matrix the all-zero rows always occupy the last locations. If the row inter-changing takes place then the equations in the reduced system may not necessarily occupy the original positions. To avoid putting the constraints on the wrong displacements, a track of all the original equation numbers during the elimination must be kept. In our program this is done by creating a one-dimensional integer array $IEN(2N)$, which stores the original equation numbers. In the event that two equations are interchanged, the corresponding entries in IEN are interchanged as well. Subsequently, after the displacements are computed according to the methodology outlined in this section, they are rearranged back according to IEN , so they appear at the appropriate locations corresponding to the original vector.

7.3.6 Determination of Contact Forces

The contact forces among the disks are equal to the forces in the corresponding truss elements of the equivalent truss system. The forces in the trusses depend on the displacements of the corresponding nodes that span the elements.

The methodology of calculating the nodal displacements for the equivalent truss system was presented in Section 7.3.5. As was pointed out in that section, the equivalent truss system, in a vast majority of situations, will not be in static equilibrium, and therefore, typically, a number of additional constraints on displacements (supports) will be added to the system. If such is the case, then it may appear to the reader that the given structure can be constrained in a number of different ways. In other words, the required constraints can be added to the system at a variety of different places, just as the set of linearly independent equations can be selected from the global set in a number of different ways. The reader may therefore argue that, considering the method used in finding the set of linearly independent equations, a different numbering of the disks could result in different constraints being added, and hence, a different computed nodal displacement vectors. This, of

course, is true. The displacements as computed in Section 7.3.5 represent the absolute nodal displacements experienced by the structure which is constrained in a certain way. If the given structure is constrained in a different way, then the corresponding absolute nodal displacements will, inevitably, be different as well. However, the relative displacements among the nodes will remain unchanged. In other words, regardless of where the required constraints are added to the structure, as long as the resulting system is in static equilibrium, the displacements of the nodes relative to one another will remain the same. The relative displacements amongst the nodes are the measure of the internal stiffness or rigidity of the given structure, which in turn, depends on the arrangement of the structural elements comprising it, and not on the particular way in which the structure is fixed in space. The method with which the nodal displacements are computed does not affect the rigidity of our equivalent truss system, but it merely restricts its movement in the directions corresponding to the system's degrees of freedom, or simply "fixes" it in space. Consequently, since we are only interested in the internal forces of the truss elements, which depend on the relative nodal displacements of the system, it can be concluded that the proposed method of computation is justified.

We will now proceed with calculation of the forces in the truss elements. A typical truss is shown in Figure 7.9 in Section 7.3.2. We recall that in local coordinate system a truss element can only experience a change of length. Hence, similarly to Equation (7.14) we may write an expression for the force in a truss,

$$p_i^j = k_{ii}^j \cdot \delta_i^j + k_{ij} \cdot \delta_j^i \quad (7.40)$$

Using Equation (7.27) and Equation (7.37) the above can be written as

$$p_i^j = \frac{2E}{\pi} \cdot (\delta_i^j + \delta_j^i)$$

But since the local displacements can be related to the global displacements through the coordinate transformation matrix, we may write,

$$p_i^j = \frac{2E}{\pi} \cdot ([R_{ij}] \cdot [\Delta_i] + [R_{ji}] \cdot [\Delta_j])$$

and by Equation (7.28) and Equation (7.21) this becomes

$$p_i^j = \frac{2E}{\pi} \cdot [l \ m] \cdot \left(\begin{bmatrix} \Delta_x \\ \Delta_y \end{bmatrix}_i - \begin{bmatrix} \Delta_x \\ \Delta_y \end{bmatrix}_j \right) \quad (7.41)$$

where coefficients l , m , in the above equation are defined by Equation (7.29).

The components of the nodal displacements can be obtained from the computed global displacement vector according to Equation (7.36). The determination of the contact forces now becomes straight forward and can be summarized as follows:

1). Contact forces among disks.

If disk i and disk j are in contact, i.e. $IC(i,j) = 1$, then:

$$L_{ij} = R_i + R_j, \quad l = \frac{X_i - X_j}{L_{ij}}, \quad m = \frac{Y_i - Y_j}{L_{ij}}$$

and

$$p_i^j = \frac{2E}{\pi} \cdot [l (\Delta_{2i-1} - \Delta_{2j-1}) + m (\Delta_{2i} - \Delta_{2j})]$$

2). Contact forces between the disks and the obstruction

If disk i and obstruction line segment j are in contact, i.e. $IC(i,j) = 1$, then:

$$L_{ij} = R_i, \quad l = \frac{X_i - X_j}{L_{ij}}, \quad m = \frac{Y_i - Y_j}{L_{ij}}$$

where X_j, Y_j are the coordinates of the point of contact, and

$$p_i^j = \frac{2E}{\pi} \cdot [l \Delta_{2i-1} + m \Delta_{2i}]$$

This concludes the computation of the contact forces. For our choice of the local systems of coordinates the positive contact forces will indicate the tension between the disks, while the negative contact forces will correspond to compression.

7.4 CONCLUSIONS

The method of determination of the contact forces, presented in this chapter was programmed and successfully incorporated into our model. The method consists of two major parts: dynamic and static analysis.

The dynamic analysis is based on Newton's Second Law of motion. It is easy to implement and requires relatively low computational times. The technique, however cannot be used to determine the contact forces for disks having more than two points of contact.

The static analysis is based on a well known technique of structural mechanics, often referred to as the matrix structure analysis or stiffness method. The introductory theory for this technique was discussed briefly in Section 7.3.1. More detailed description of the theory can be readily found in many textbooks on structural analysis, [3],[8]. The implementation of the methodology presented in Section 7.3.1, to analyze the equivalent truss systems which replace the disk conglomerates was presented in Section 7.3.2. The technique developed is very versatile and can be used to determine the contact forces among any conglomerates or clusters of disks, regardless of whether they are rigid or not, or whether they

are constrained or unconstrained. Moreover this technique can be used to calculate the contact forces among the disks with one or two points of contact. Consequently, one global system of static equilibrium equations could be set up for all the disks present in the system, from which the contact forces could be computed. In other words, static equilibrium analysis alone could be used for this purpose. The versatility of the static analysis method in handling any type of disk conglomeration and the disks with any number of points of contact, has, however, its drawback. The problem is that the method involves a large number of computations, summations and multiplications, which is of the order of $(2N)^4$, where N represents the number of disks in the system. Henceforth, a seemingly redundant and unnecessary dynamic analysis becomes not so in terms of the computational time involved.

The dynamic method is very fast in comparison to the static one, and can quickly reduce a given system of disks into a much smaller sub-system to which the static analysis can then be applied and the remaining contact forces computed.

The verification of both methods was performed. Disk conglomerates for which the contact forces could be determined by either method, were used in the verification procedure. The contact forces computed by both methods were the same.

CHAPTER 8

APPLICATIONS

8.1 INTRODUCTION

A computer simulation program has been developed which allows a user, through the choice of the initial parameters, configuration of the system's boundaries, and the external force routine, to model virtually any two-dimensional multi-body systems. This chapter will discuss some of the foreseeable applications. It is not the purpose of the present thesis to go in depth to understand the nature of the internal interactions, external forces present, and the parameters, of any specific system that can be modelled. Therefore, in this chapter we shall limit ourselves to indicating various possibilities for this model and presenting the visualization of the process that the chosen system is undergoing.

Although, as indicated earlier, this model is very general and can be applied to virtually any planar multi-body system, the main purpose for its development was its use in modelling the motion of the broken ice in open water as well as in rivers and channels. The very nature of the processes

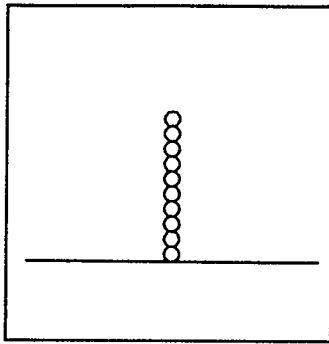
involving the broken ice such as jam-ups, ice accumulation, non-homogenous and time dependant ice surface density, and inherent randomness of the shapes, sizes, and frequencies of appearances, prevent models based on the continuum theory from being applied successfully. It is believed that the present model, which is discrete and thus not hindered by the problems facing continuum models, can be used successfully in analyzing the motion of broken ice.

An application for the model, which naturally suggests itself, is the flow of granular material in chutes and on conveyors. This possibility is also contemplated here.

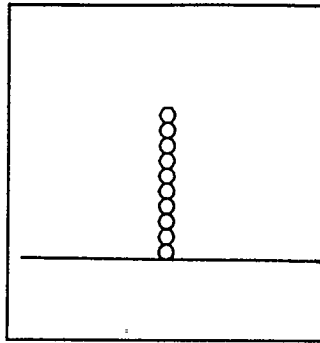
The motion of orbiting bodies, whose trajectories may change with time, is another possible area in which the model could be applied.

Before proceeding with the presentation of different examples of application, a verification of the accumulation of the numerical errors is done. For this we used ten disks of equal radius, arranged to form a vertical stack, as shown on the next page. The only external forces specified were the gravity forces (acting vertically downwards), and all the initial velocities were set to zero.

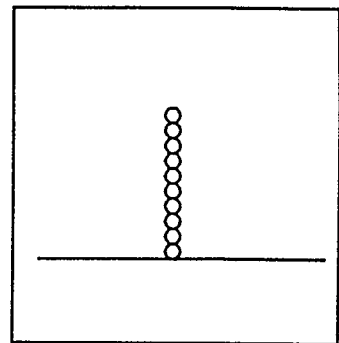
Such a configuration of disks is clearly in static



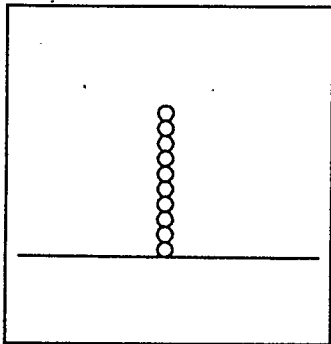
$t = 0.8 \text{ sec.}$



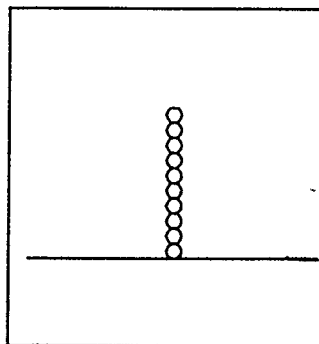
$t = 1.6 \text{ sec.}$



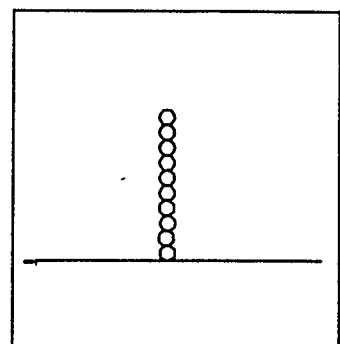
$t = 2.4 \text{ sec.}$



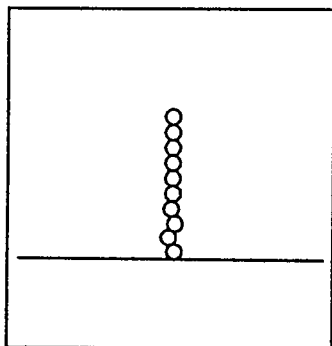
$t = 3.2 \text{ sec.}$



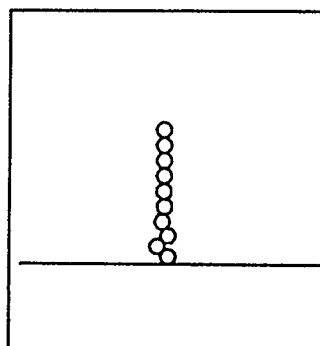
$t = 4.0 \text{ sec.}$



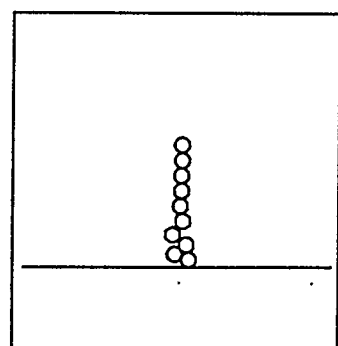
$t = 4.8 \text{ sec.}$



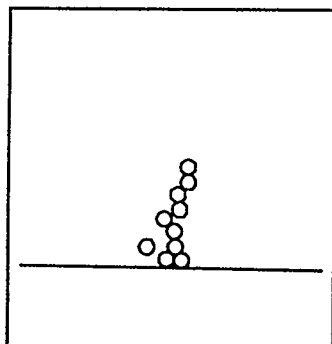
$t = 5.6 \text{ sec.}$



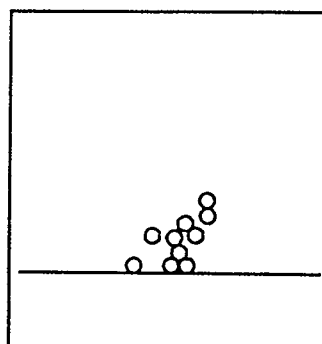
$t = 6.4 \text{ sec.}$



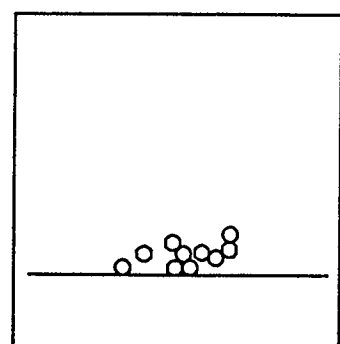
$t = 7.2 \text{ sec.}$



$t = 8.0 \text{ sec.}$



$t = 8.8 \text{ sec.}$



$t = 9.6 \text{ sec.}$

equilibrium and it should, ideally, remain so throughout the simulation, since the initial velocities are zero and the "x" components of the external forces acting on the system are zero.

The accumulation of the numerical error during the computer simulation provides a "necessary disturbance" to create an instability in a system such as this. Over the many iterations during which the equations of motion are generated and integrated, the small numerical error will accumulate sufficiently to give rise to a small change of the transverse coordinate of the center of one of the disks. Consequently, the position of the disk shifts away from the vertical line of equilibrium and since the total force on it is no longer along the corresponding constraint, a small downward acceleration develops and the entire system becomes unstable.

In the example shown here the time step used was 0.001 seconds. The program was executed in single precision arithmetics. The instability developed at about 5 seconds, which is equivalent to 5000 time steps. Such level of accumulation of numerical error, considering the complexity of the processes that this model is intended for, is viewed as acceptable.

8.2 MOTION OF BROKEN ICE

As will be seen in this section the program shows a great potential for modelling the motion of broken ice in open waters as well as in the rivers and channels. There is a great flexibility in generating the ice floes and the creation of obstructions which allows a variety of scenarios to be considered. For example, a complicated shape of a river bank could be reconstructed, and the ice floes could be generated at random according to some observed distribution. Also, to this end, random shape ice floes could be created from disks (Section 4.6). This would give a great similarity to the actual process being modelled.

Two examples will be presented in this section to demonstrate the model's applicability to both rivers and channels, and to open water.

8.2.1 Rivers and Channels

There are two instances during the yearly cycle of river temperature regimes, when the flow of isolated ice bodies occurs. In the fall the freezing process is accompanied by the flow of small ice formations (ice pans), which leads to subsequent ice accumulation and eventual freeze-up. In the spring the melting of the ice cover leads to break-up and the ice run. The present model is directed towards the analysis of systems of isolated bodies, and it is thus believed that it can be applied to study the phenomena associated with river ice transportation, such as rate of ice accumulation around obstructions, formation of ice jams, or bridging.

As a numerical example a part of the Liard river geometry is considered, as shown in Figure 8.1 below. The banks of the river were discretized using 58 line segments.

The ice floes were generated at the entrance to the control area, located on the left side. The velocity of the stream at this point was assumed to be 2m/s. Thereafter, the flow field over the entire control area (within the solid boundaries) was calculated using the computer routine of Chapter 5. Ice floes of circular shape were randomly generated using the routine of Chapter 4. It was assumed that

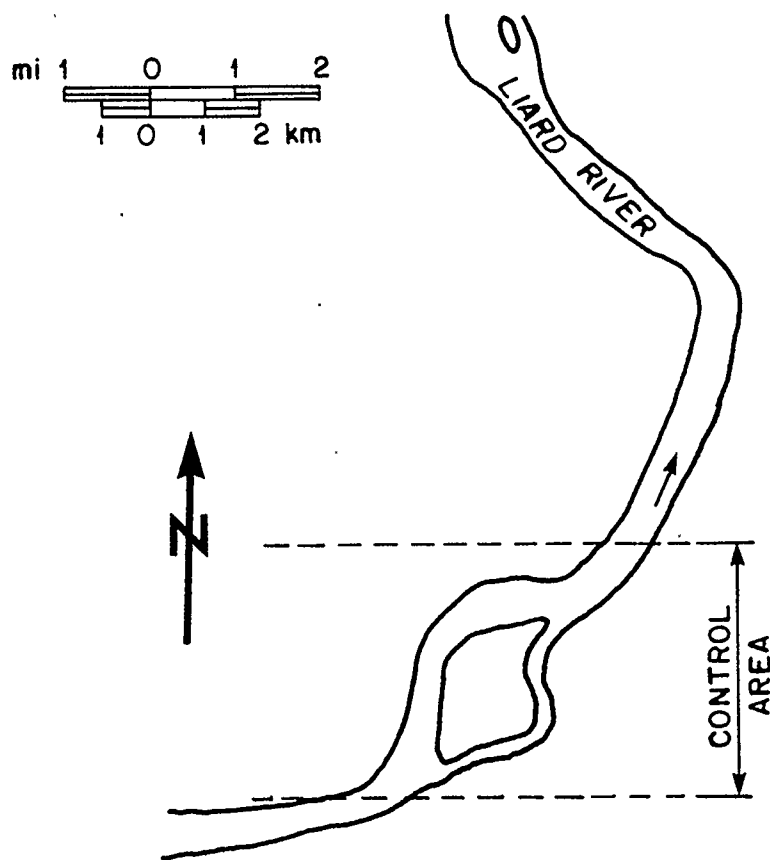
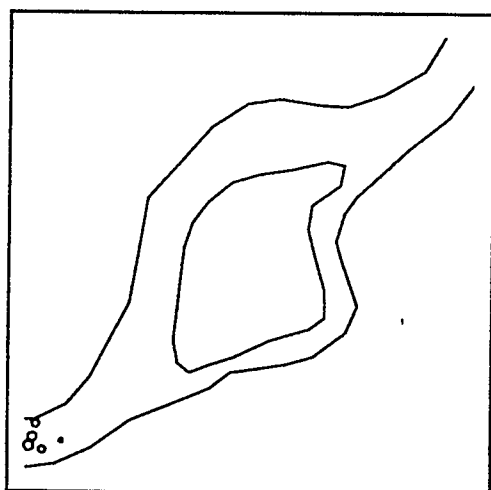
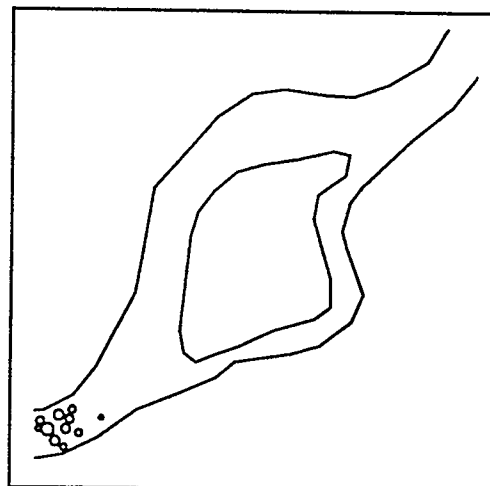
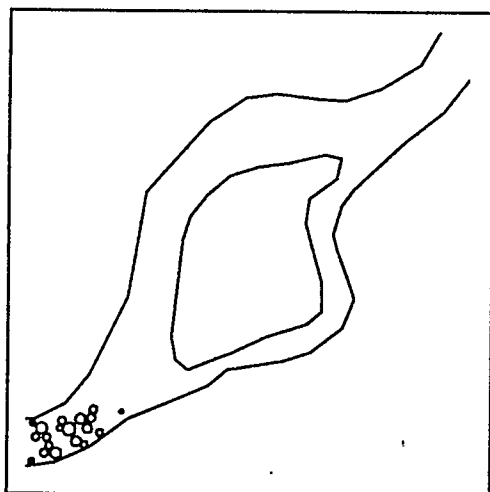
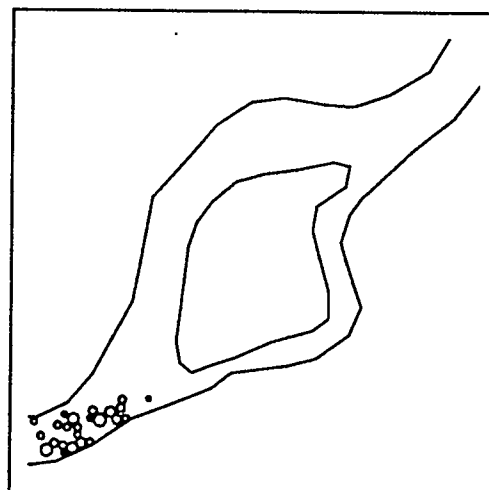
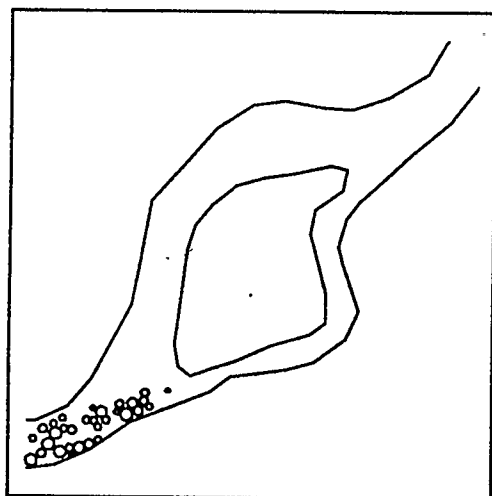
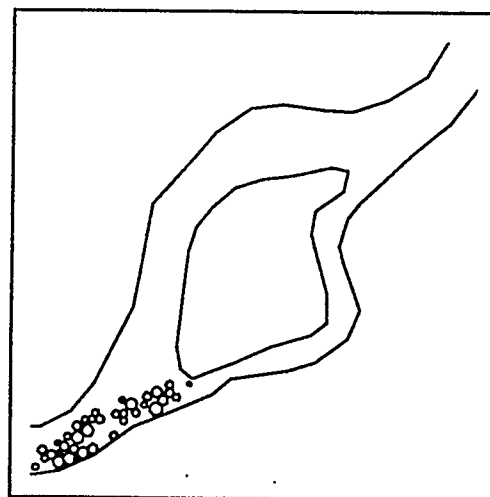


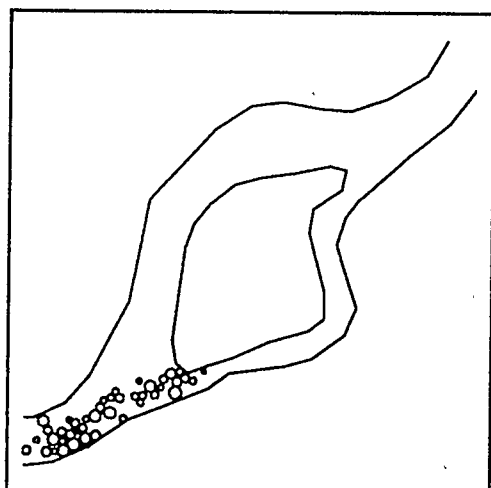
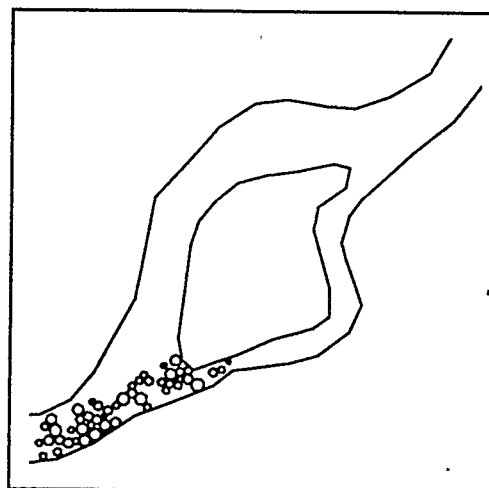
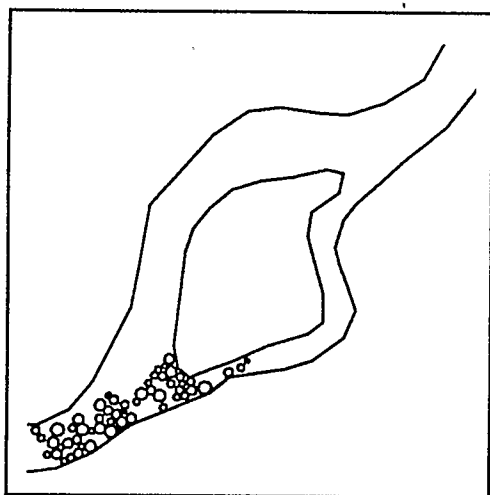
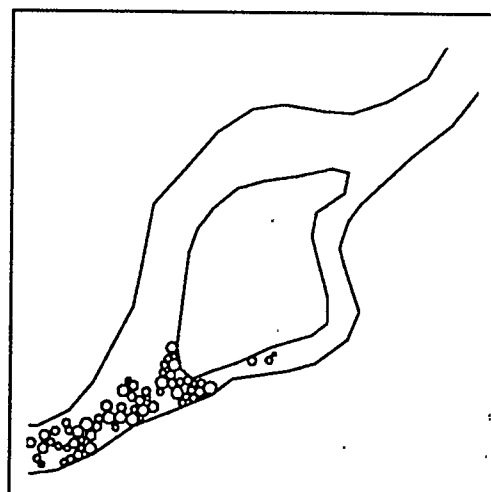
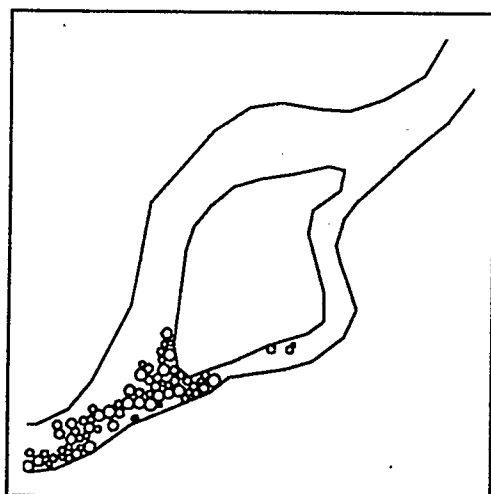
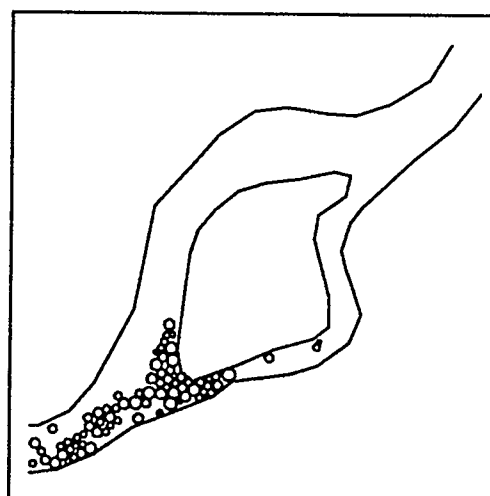
Figure 8.1. Control area of the section of Liard river.

the radii of the ice floes were distributed according to the Gaussian distribution with the following parameters:

$$\mu = 55\text{m} , \quad x_{\max} = 80\text{m} , \quad x_{\min} = 30\text{m} , \quad \sigma = 15\text{m} .$$

It was also assumed that the frequency with which the floes enter the control area is given by a Poissonian distribution which parameters were determined (Section 4.5) from the value of the ice surface density ρ . A sample run was conducted with $\rho = 0.345$ and the results are shown on the next two pages.

 $t = 100 \text{ sec.}$  $t = 300 \text{ sec.}$  $t = 500 \text{ sec.}$  $t = 700 \text{ sec.}$  $t = 900 \text{ sec.}$  $t = 1100 \text{ sec.}$

 $t = 1300 \text{ sec.}$  $t = 1500 \text{ sec.}$  $t = 1700 \text{ sec.}$  $t = 1900 \text{ sec.}$  $t = 2100 \text{ sec.}$  $t = 2300 \text{ sec.}$

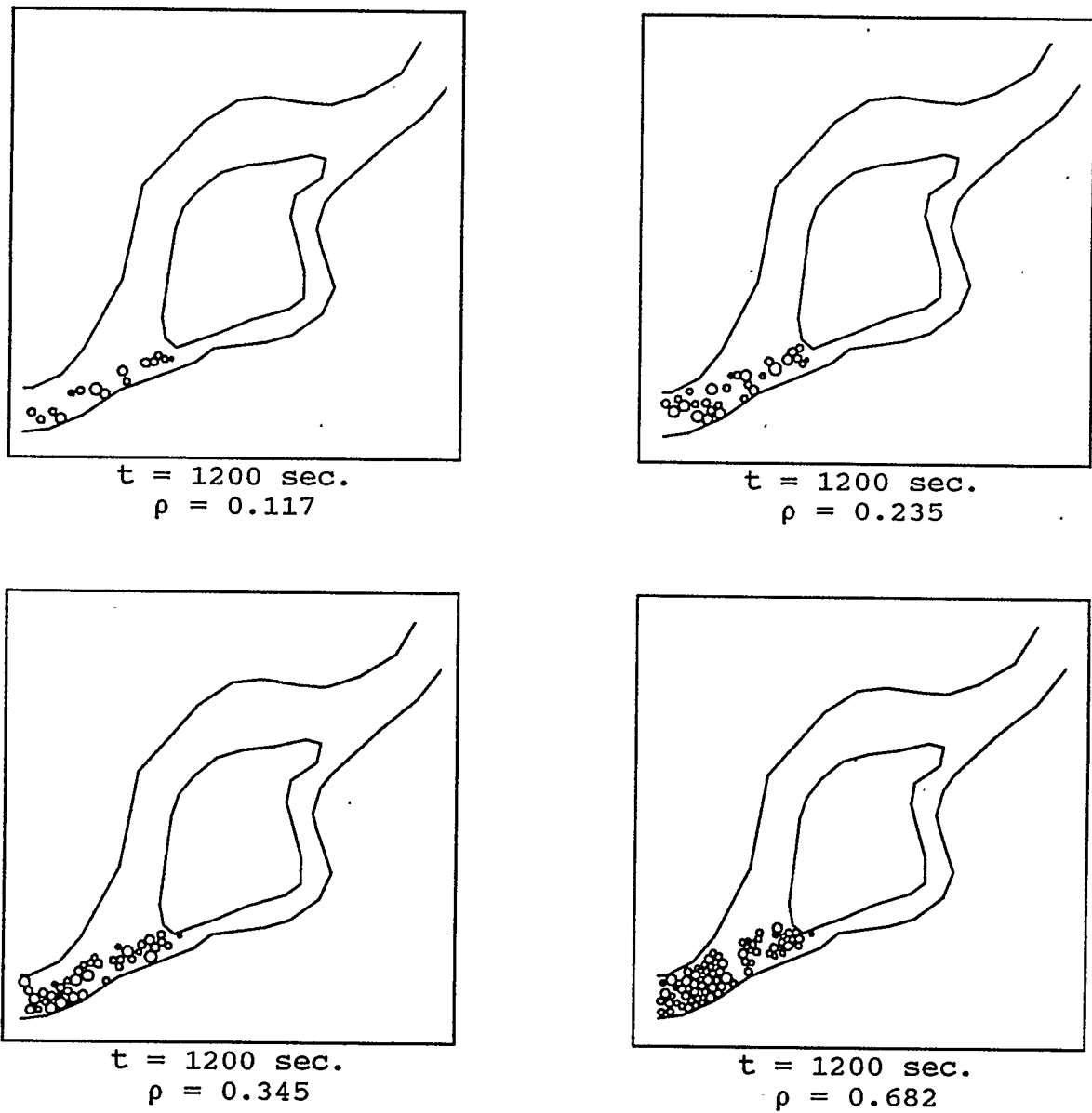


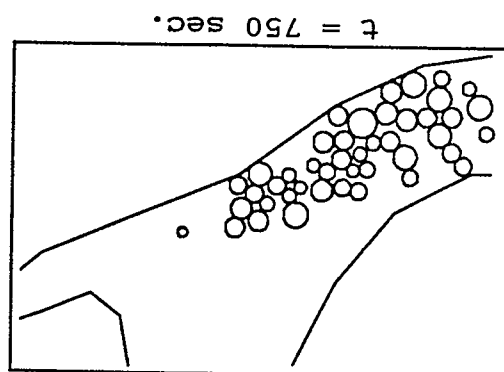
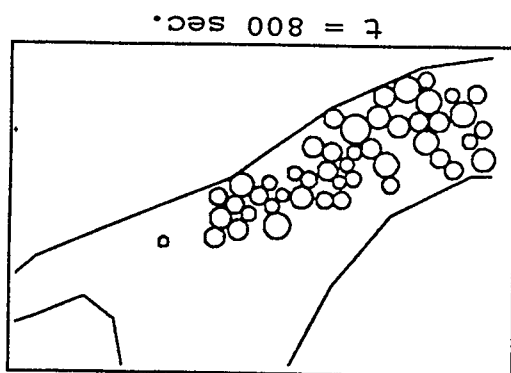
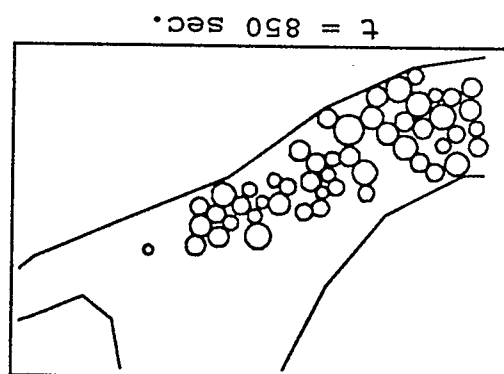
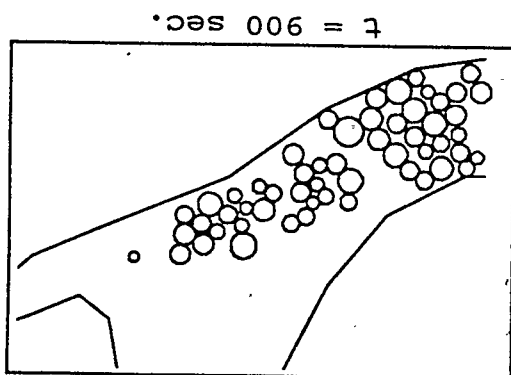
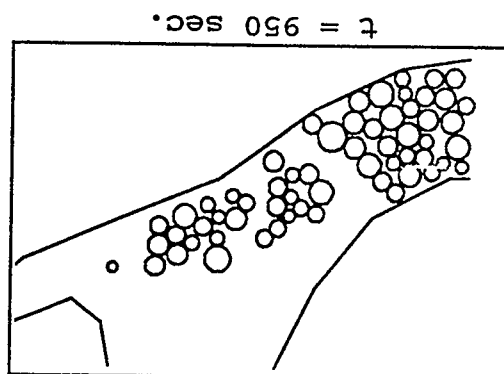
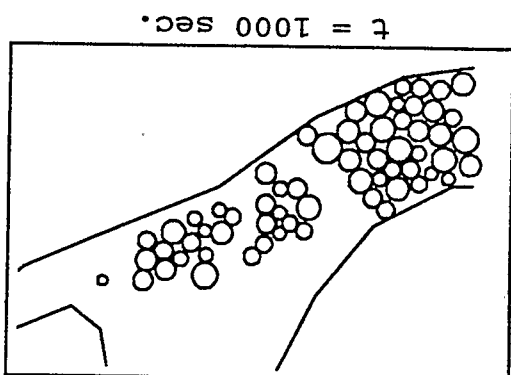
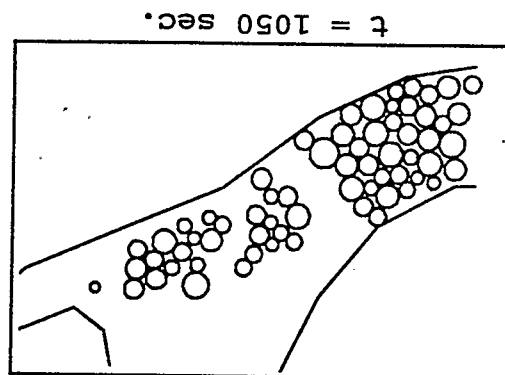
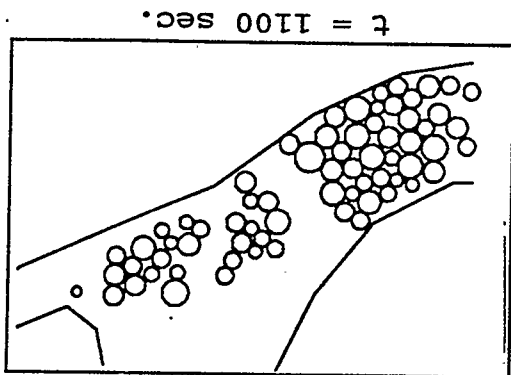
Figure 8.2. An illustration of various surface densities.

In the example shown in Figure 8.2 simulation was carried out for four different surface densities with the same parameters as in the previous example. The surface densities used are indicated in the Figure. At low ice concentrations the individual floes move with the mean current velocities

[1]. At high concentrations of ice (high surface density) the motion of the floes is mostly governed by the ice-ice interactions. Consequently, the increased clustering occurs and the motion of ice does not necessarily comply with the mean flow. The results of the numerical simulation performed seem to be in general agreement with the above statements.

At high levels of surface concentrations of ice, the interactions among the floes and the walls of the channel may result in sufficiently large contact forces (or stresses) to greatly hinder, or even (at sufficient concentrations of ice) prevent the movement of the surface ice layer. It is believed that the present model can be used to model the occurrences of such no-flow conditions. An illustration of this is shown on the next page. The section of Liard river was considered and the surface density ρ was increased to 0.68. At a time of about 1000 sec. into the simulation a considerable clogging-up of the channel took place, resulting in a significant reduction of the surface discharge of ice. A jam-up (no-flow condition) was not observed as the channel is diverging at this particular place.

It should be pointed out, that the mathematical model is restricted to two dimensions and thus can only describe the flow on the water surface. Consequently, only the initiation of an ice jam can be simulated with this model, since after



that instant in time ridging frequently occurs and the ice system becomes three-dimensional.

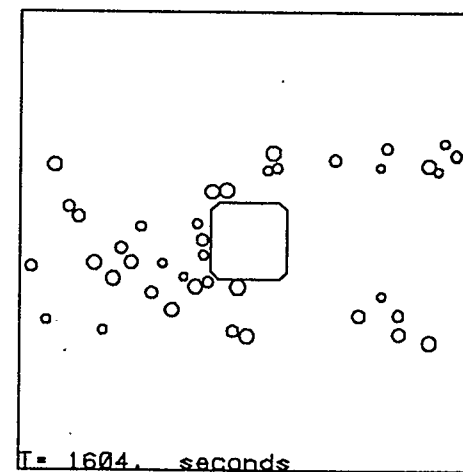
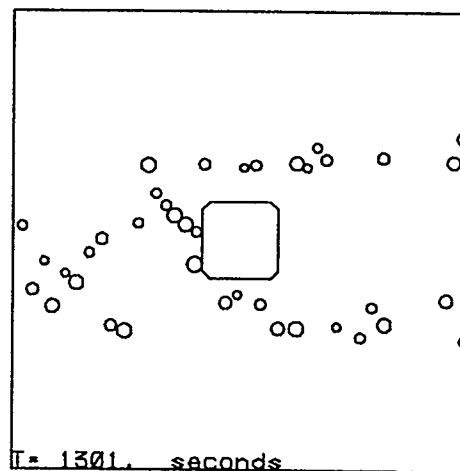
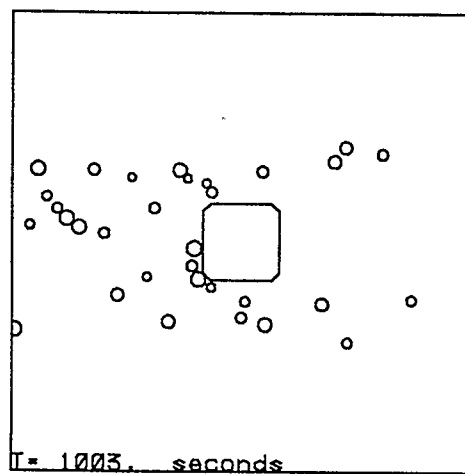
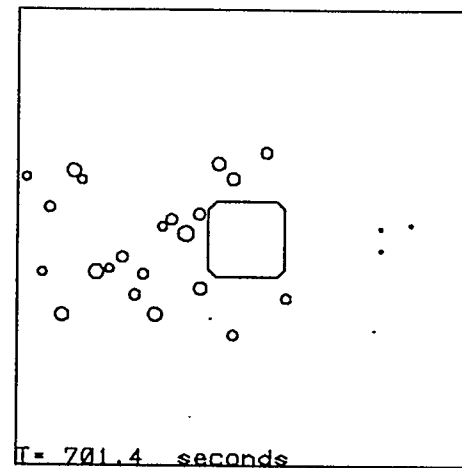
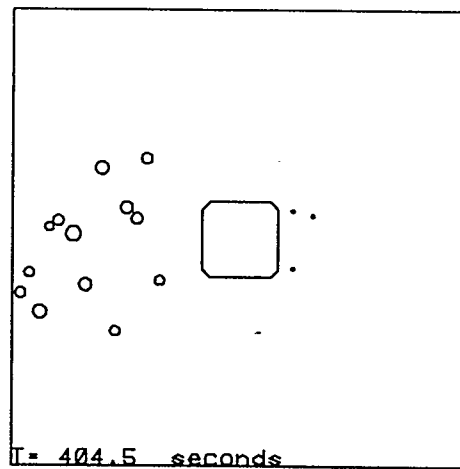
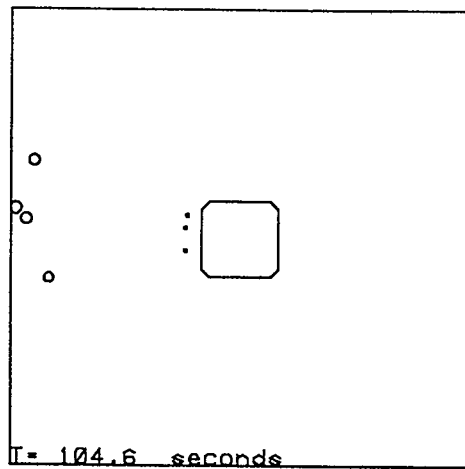
8.2.2 Open Water

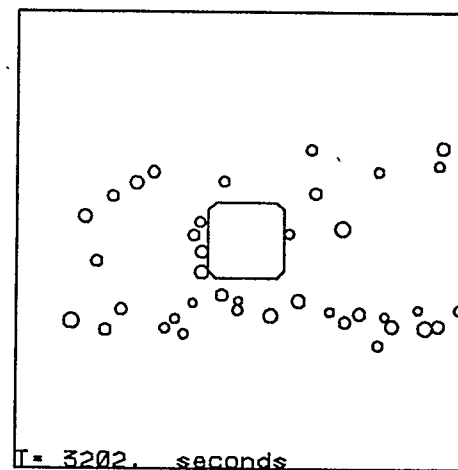
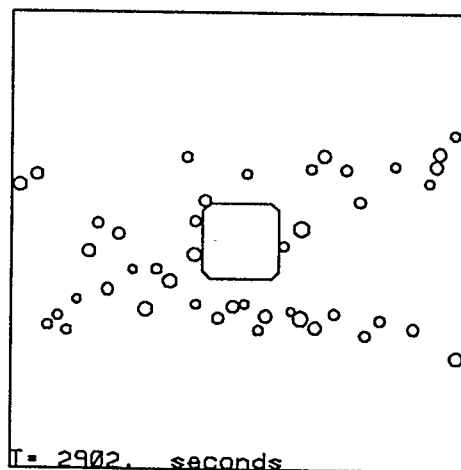
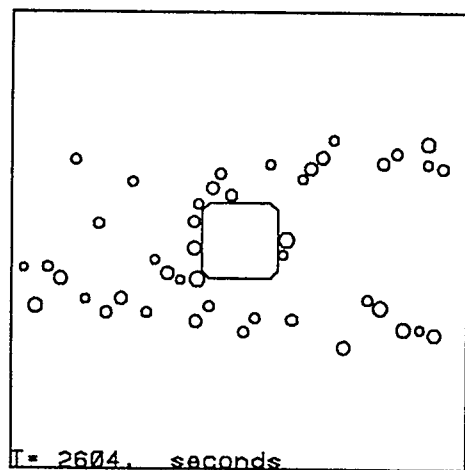
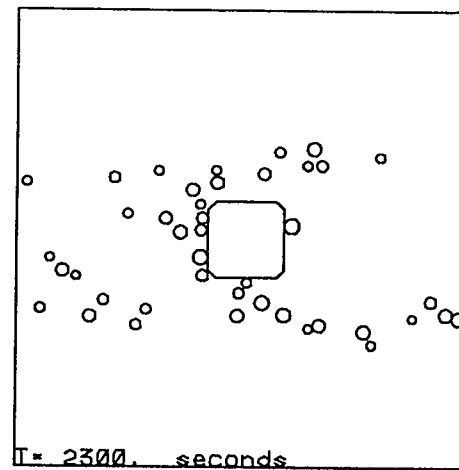
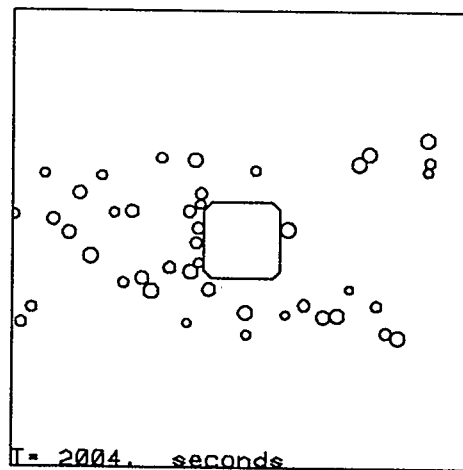
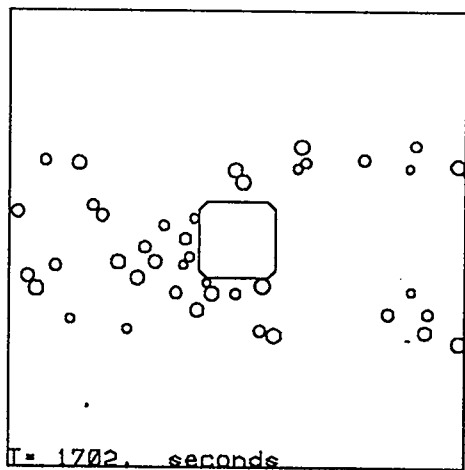
Two computer runs were carried out to illustrate the applicability of the model to analyzing the motion of broken ice in open water or an ice-off-shore structure interaction. The structure was assumed to be of a square shape with cut-off corners. The free stream velocity was set to be in the positive x direction with a magnitude of 0.5m/s. The ice floes were assumed to be distributed according to the Gaussian distribution with the following parameters:

$$\mu = 7.5\text{m} , \quad \sigma = 1.5\text{m} , \quad x_{\text{max}} = 10\text{m} , \quad x_{\text{min}} = 5\text{m} .$$

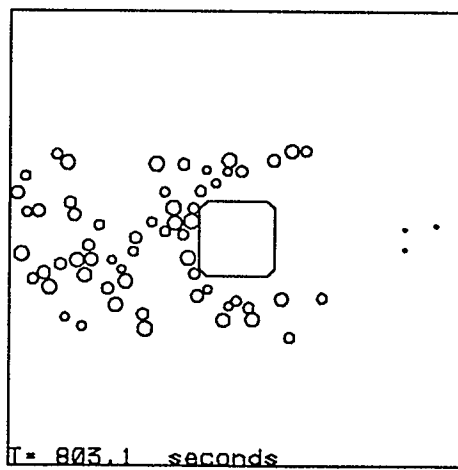
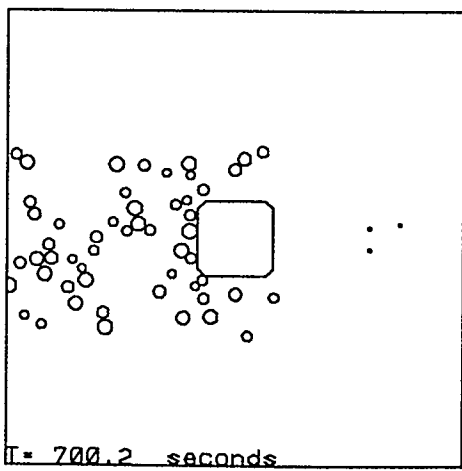
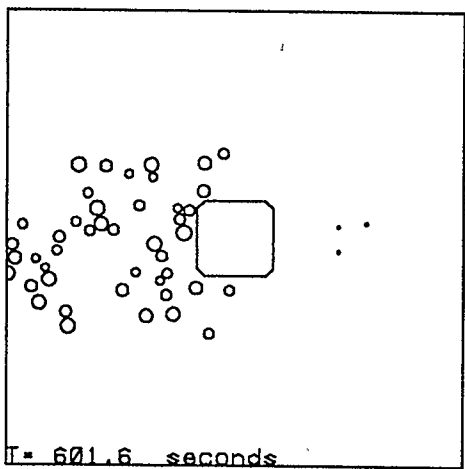
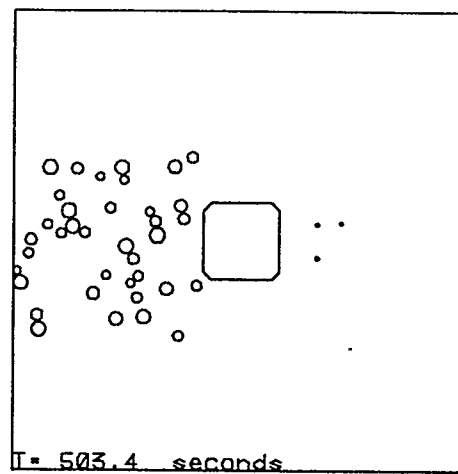
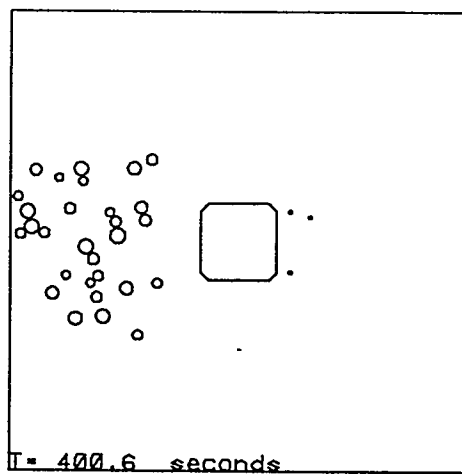
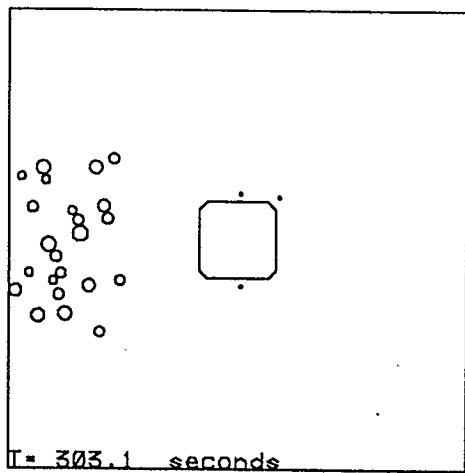
The coefficient of restitution was chosen to be 0.4, and the coefficient of friction was selected as 0.5. The ice floes were assumed to be circular in shape. The computer routine of Chapter 5 was used to determine the stream velocity field around the structure.

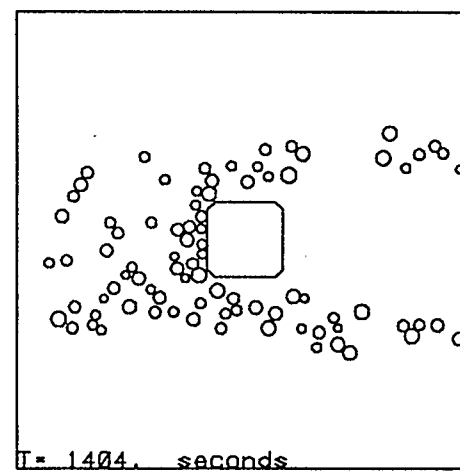
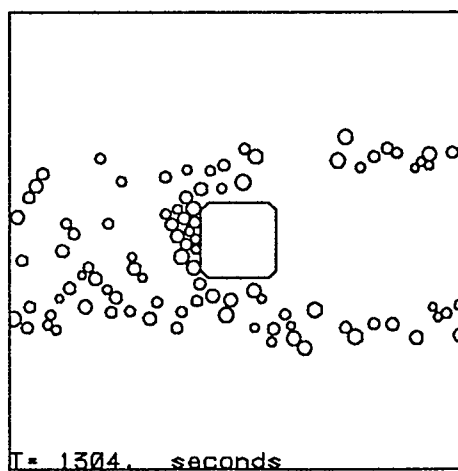
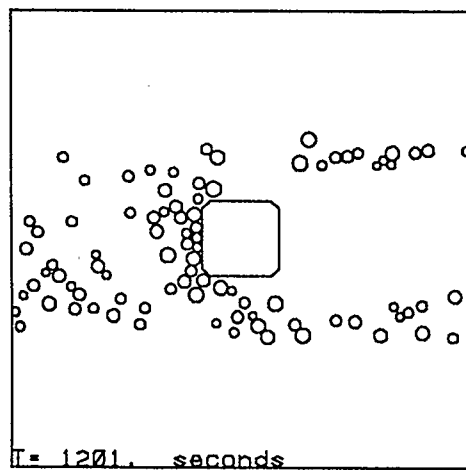
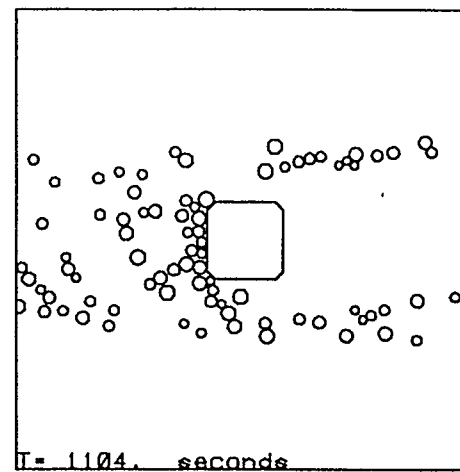
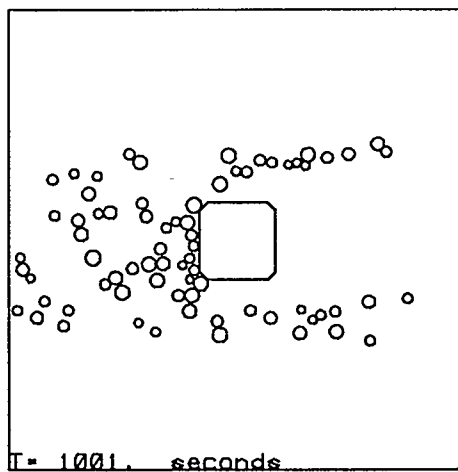
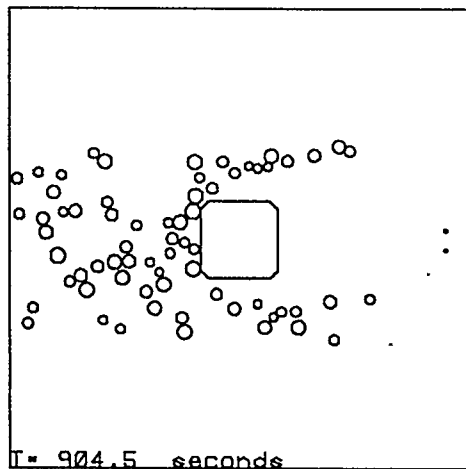
The first run was conducted with a relatively low surface density ρ , of 0.05. The computer print-out of this instance is shown on the next two pages. The simulation was carried out over an extended period of time. No significant accumulation of ice (with time) around the structure was observed, which would suggest that low ice densities should not pose a problem for off-shore structures.





The second computer run was carried out with the ice surface density ρ of 0.15, i.e. three times higher than that in the previous example. The results from the run are shown on the next two pages. In this instance a steady increase of the number of ice floes attached to the structure was observed, which could suggest the possibility of the accumulation of ice around the structure.





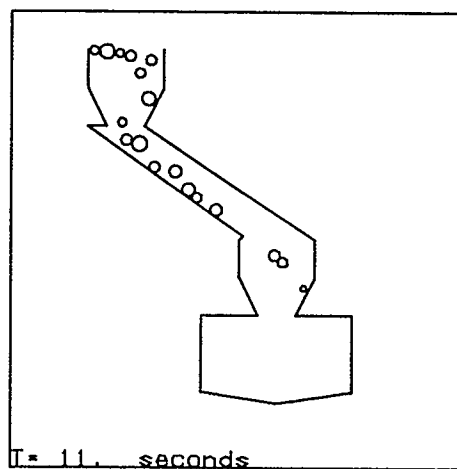
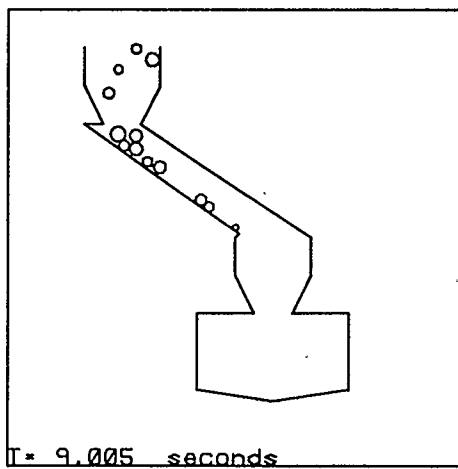
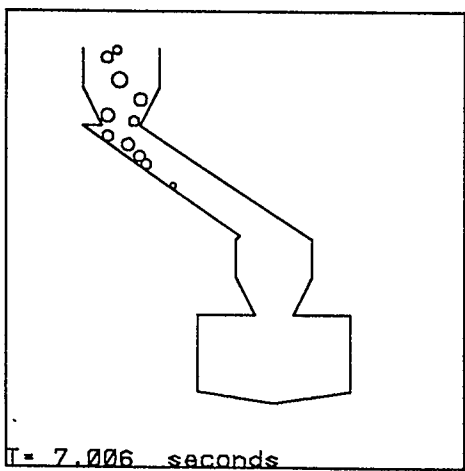
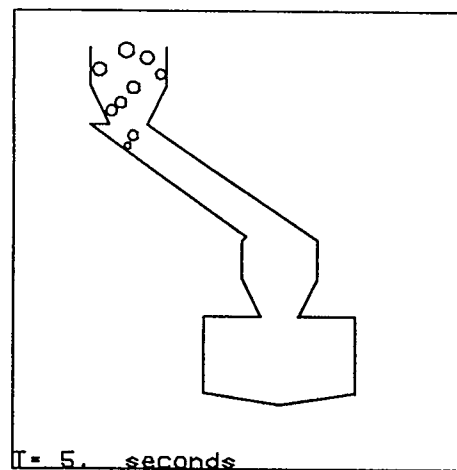
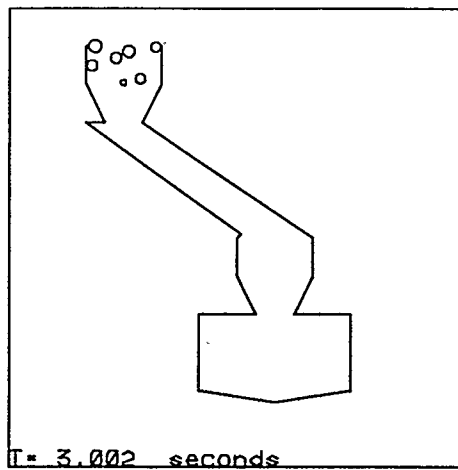
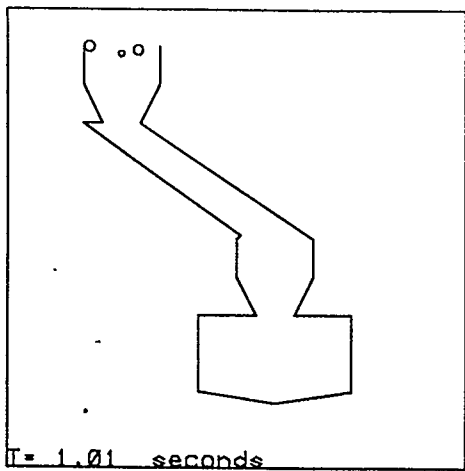
8.3 MOTION OF GRANULAR MATERIAL

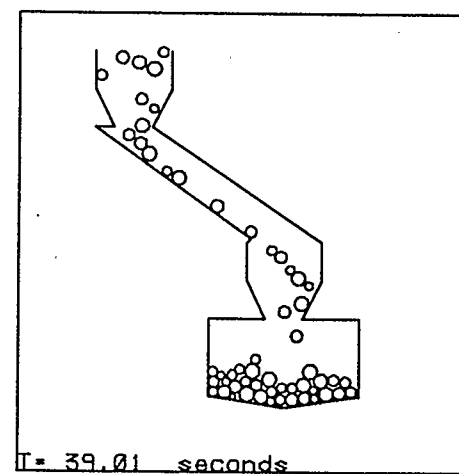
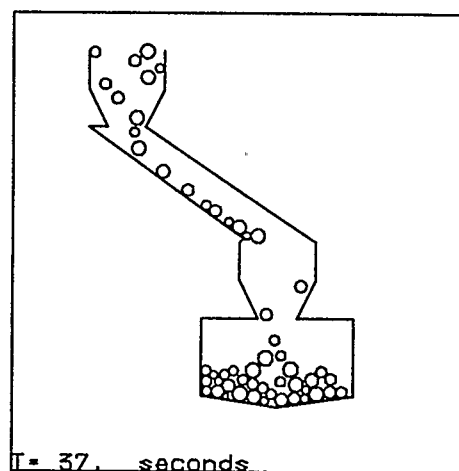
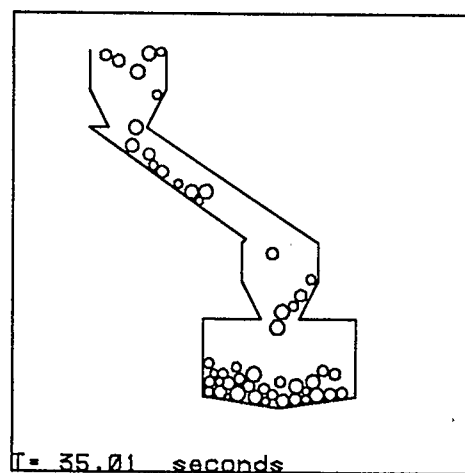
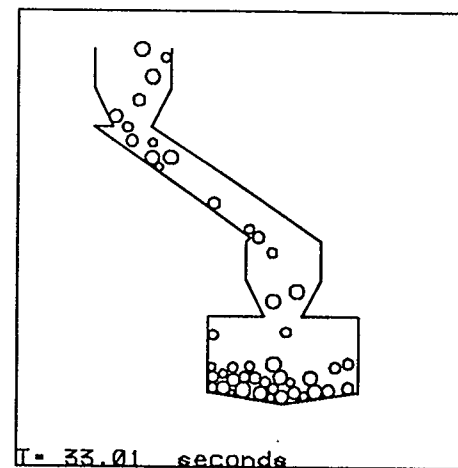
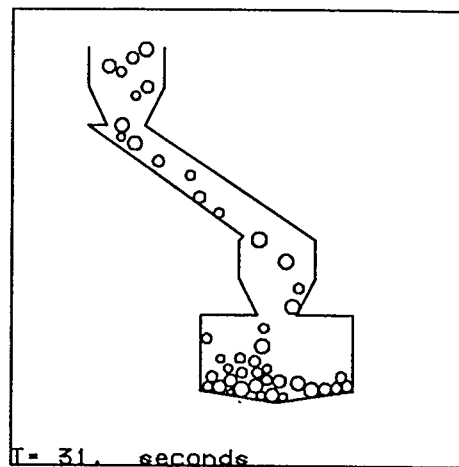
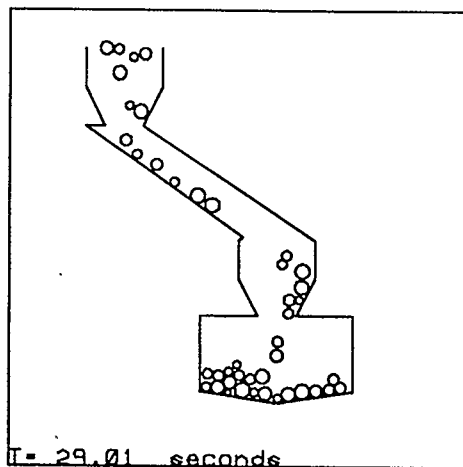
In the following example the model is used for analyzing the filling of a bin with dry cohesionless granular material travelling through a chute. The individual granules are represented by disks. The external forces consist of only gravity. The rigid bodies were generated at the top of the chute with average rate of arrival λ equal to 1.75/sec. The size distribution of the rigid bodies was assumed to be Gaussian with the following parameters:

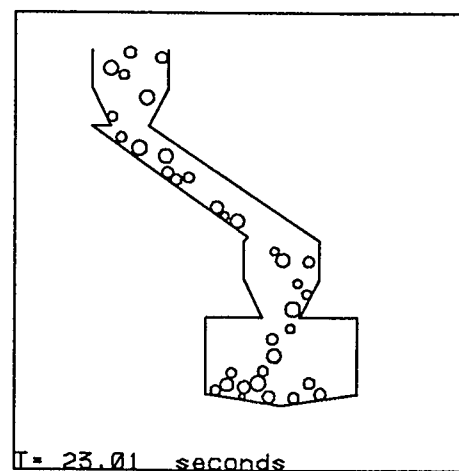
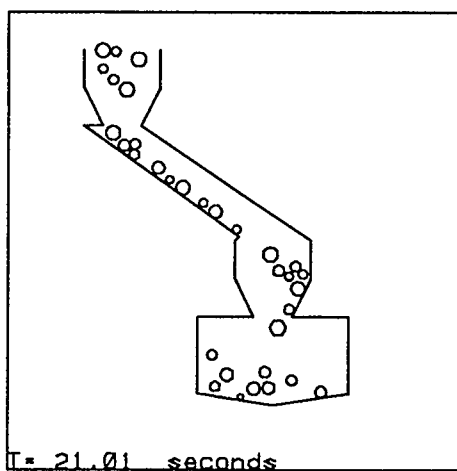
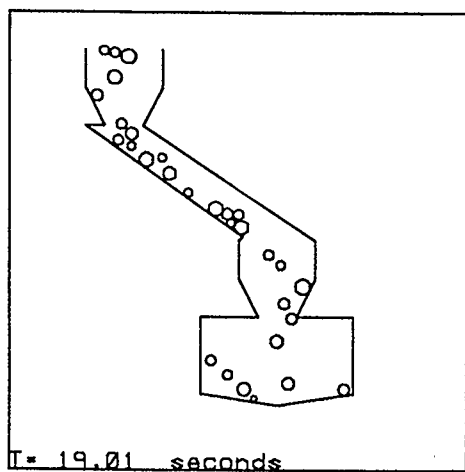
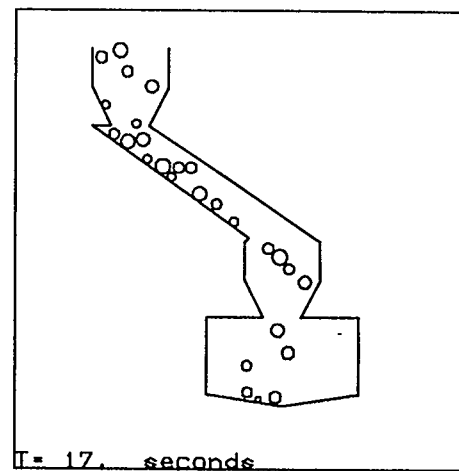
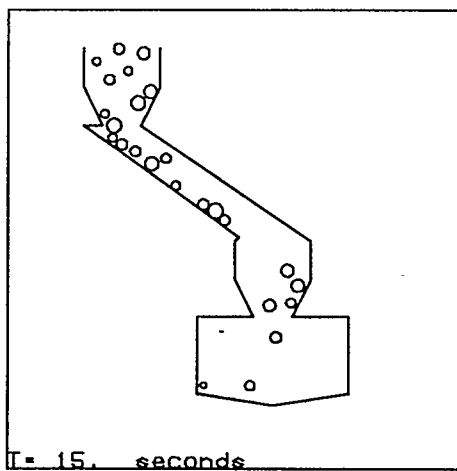
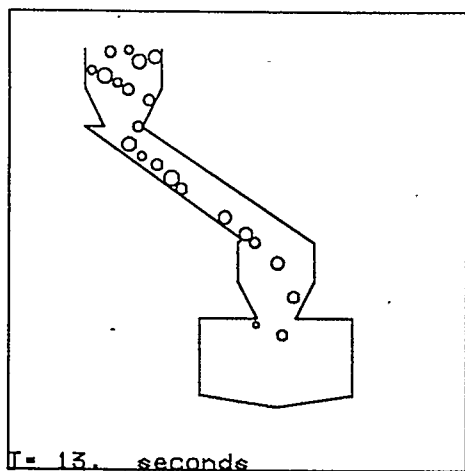
$$\mu = 7.5\text{cm.} , \quad x_{\max} = 10\text{cm.} , \quad x_{\min} = 5\text{cm.} , \quad \sigma = 1.5\text{cm.}$$

The size of the bin was 200cm. wide and 100cm. high. The results are indicated on the next three pages.

The issue that was of primary interest here was the demonstration of the applicability of the model to investigate the material transport capacity of such systems. The graphical data shown indicates that the chute-bin system used in this example was capable of transporting the material at the given rate of 1.75/sec. .







8.4 CONCLUSIONS

In this chapter various possible applications of the computer model were illustrated. The present thesis deals with the development of the general simulation methodology, rather than with a detailed analysis of any particular system, and thus no quantitative data, such as the distribution of stresses within the granular material or the forces exerted on the solid boundaries, was presented. This type of undertaking would require a prior detailed analysis of the given system and the character of the internal and the external forces present, which itself in many instances represents a considerable challenge.

The main purpose of this chapter was to show the variety of different systems and situations that can be simulated, as well as the versatility of the developed model. It is believed that this was accomplished. It should also be pointed out that all the tools necessary for obtaining quantitative data are in place, as they comprise a necessary part of computations. For example, the contact forces among the rigid bodies, or the loads exerted by the rigid bodies on the solid boundaries (walls of a channel, or an obstruction) are computed at every time step during the simulation, which in turn can be translated into the shear or normal stresses.

Moreover, various checks were performed on the model, and it was concluded that the model is dynamically correct, that is, that it complies with the basic laws of mechanics such as Newton's second law or the conservation of momentum.

Also, as illustrated in Section 8.1 a test was performed on the amount of numerical error accumulation. The test involved ten disks and one obstruction line segment. The error accumulation was at acceptable level despite the fact that only single precision arithmetics were used.

CHAPTER 9

CONCLUSIONS AND RECOMMENDATIONS

9.1 CONCLUSIONS

A general simulation methodology for analyzing two-dimensional topologically variable multi-body systems has been developed. The mathematical model and the resulting equations of motion have been derived from the principles of Lagrangian dynamics. A motion of a system is described by a set of differential equations of motion and a set of algebraic constraint equations. The mathematical formulation is exact since no simplifying assumptions were made during the formulation. Disks and straight line segments have been chosen as the two primary elements to represent the actual systems. With these any shape of solid boundaries and rigid bodies can be described. Overall the methodology offers a great versatility in modelling a variety of multi-body systems with either one-sided or double-sided constraints.

The main accomplishments of the present investigation can be summarized as follows:

1. A concept of static and dynamic analysis has been introduced to handle the one-sided constraints. It was proposed that the constraints (connections) remain intact throughout the simulations until the time at which the constraint forces become tensile. Thus the constraint is replaced by the assumed dependence among the generalized coordinates, which in turn allows for a reduction in the number of the generalized coordinates and in the number of constraint equations. Consequently, the unknown constraint forces cannot be computed from the constraint equations and must be determined by other means. This is accomplished through the dynamic analysis (based on the Newton's second law) and the static analysis (based on finite element truss system analysis).
2. An algorithm to perform the above analysis on any arbitrary multi-body system has been developed and successfully implemented on a digital computer.
3. A method of handling the inter-body friction was introduced and numerically implemented. The difficulty in handling friction lies in the fact that the friction forces depend not only on the unknown constraint forces but also on the unknown orientations of the relative velocities across the constraints. Whence, the equations of motion which include frictional effects, cannot, in

general, be generated for variable topology multi-body systems. The proposed technique solves for the frictional forces independently of the solution for the generalized coordinates. This in effect makes the frictional forces "lag" one time step behind all other parameters. The numerical errors thus introduced are not considered significant as, at the most, these represent second order terms with respect to time.

4. A method of fluid flow calculation has been established that allows for the calculations of the drag forces on the rigid bodies. The method is based on the potential flow theory. The flow field is governed by the Laplace equation with Neumann boundary conditions. The solution to the field equations is obtained through the so-called panel method.
5. The above method has been successfully implemented for both channel flow and flow around an obstruction. The panels are obtained by automatically dividing the obstruction line segments. The developed method also allows for the moving boundaries to be considered.
6. A concept of random rigid body generation has been introduced. A pseudo-random number generator which gives uniformly distributed random numbers and pairs of numbers

between 0 and 1, has been employed. The sizes of the rigid bodies are randomly sampled from a specified probability density function, which can be either continuous or discrete. The generated rigid bodies are entered into the control area (control volume) at random locations and at random time intervals, so selected that the generation process resembles a Poissonian walk.

7. An algorithm for the generation of randomly shaped rigid bodies comprised of random size disks has been developed. The technique of composing random shapes is directly compatible with the process of identification of quasi-rigid body subsystems used by the model for the purpose of reducing the number of motion and constraint equations.
8. A concept of surface density of the randomly distributed bodies was introduced and the corresponding algorithm developed. This option allows for treatment of systems characterized by a certain surface density.
9. The fortran code for the above algorithms has been generated, tested, and successfully implemented.

As mentioned earlier the present computer model offers great versatility in analyzing multi-body systems. Several points can be made to demonstrate this:

- a) The type of system that is being considered can be identified by the user-supplied external force routine.
- b) The character of the system's interactions can be identified by a set of parameters such as coefficients of drag, friction, and restitution.
- c) The concentration of rigid bodies in the system can be easily chosen through the generation parameters such as mean size and the surface density.
- d) The use of a pseudo-random number generator not only provides the user with the sequences of numbers that for all practical purposes can be considered random, but also enables him to reproduce any such sequence.
- e) The random generation of rigid bodies according to any chosen probability density function permits modelling of stochastic type systems, for which the bias associated with a predefined set of rigid bodies should be avoided.
- f) The model can accommodate cohesion among the rigid bodies

by simply specifying a connection strength. Once a connection is formed, it is maintained until the tensile force in this connection exceeds the connection strength.

- g) For systems involving broken ice, the effect of adfreezing and melting can be modelled by simply varying the strength of a connection depending on the extent of time that the given connection is maintained.

Presented in this thesis is a general simulation methodology for analyzing the motion of topologically variable multi-body systems. The developed computer model is, to the best of the author's knowledge, the first attempt at investigating unconstrained multi-body systems, in which the mathematical formulation and the resulting equations of motion are based on the principles of Lagrangian dynamics. The novelty of this work hinges not on the development of new theories, but on the adoption of existing theories, and their integration into a single concise module. It is also believed that a random generation of rigid bodies is a novelty in the field, where both the continuum and discrete models deal with predefined shapes and sizes of the rigid bodies.

Finally, the limited experience in the area of dynamics of the topologically variable multi-body systems and the lack

of available software makes the present model a viable contribution to the area.

9.2 RECOMMENDATIONS

The following tasks and investigations may be carried out as an extension to the present work:

1. An experimental validation of the model. Regardless of how thorough and precise the mathematical description is, a model should be verified experimentally.
2. An inclusion of the rotational degrees of freedom for the individual disks. At present in order to reduce the number of generalized coordinates, the rotation of the individual disks was neglected. This course of action was dictated by the lack of computational power available in our department.
3. Inclusion of the effects of the rigid bodies on the fluid velocity field. This could be accomplished by replacing the rigid bodies with panels and since the velocities of each such panel would be known, the field equation could be solved to include all the bodies in the system. At present this effect is neglected since it would considerably increase the amount of computations involved.

4. Transferring the computer model to a more powerful computer. At the present the model is running on a Sun computer system, which basically is a 386-based machine. The University of Calgary has in its possession a CDC mainframe computer. Transferring our model onto that machine and utilizing the machine's parallel processing capability would enable us to not only implement the above changes, but also would provide us with much increased computational speed (by the order of about 10) and the possibility of analyzing much bigger systems or problems.
5. Extending the model to include the third dimension.

CHAPTER 10

REFERENCES

- 1 Ackermann, N.L., Shen, H.T., Ruggles, R.W., "Transportation of ice in rivers, (333-342), 1981.
- 2 Ackermann, N. L., Shen, H. H. , " Stresses in rapidly sheared fluid-solid mixtures", Journal of the Engineering Mechanics Division, ASCE, **108**, (95-113), 1982.
- 3 Allaire, P. E., "Basics of the Finite Element Method", Wm. C. Brown Publishers, 1985.
- 4 Anderson, D. A., Tannehill, J. C., Pletcher, R. H., "Computational Fluid Mechanics and Heat Transfer", Hemisphere Publishing Corporation, McGraw-Hill, 1984.
- 5 Bangold, R. A., "Experiments on gravity-free dispersion of large solid spheres in a Newtonian fluid under shear", Proceedings of the Royal Society, Ser. A, **225**, (49-63), London, 1954.
- 6 Blevis, R. D., "Applied Fluid Dynamics Handbook", Van Nostrand Reinhold Company Inc., 1984.
- 7 Burden, L. R., Faires, J. D., Reynolds, A. C., "Numerical Analysis", Prindle, Weber & Schmidt, 1981.
- 8 Cook, R. D., "Concepts and Applications of Finite Element Analysis", John Wiley & Sons, 1981.
- 9 Cundal, P.A., Strack, O.D.L., "A discrete numerical model for granular assemblies", Geotechnique, vol. 29, (47-65), 1979.
- 10 Gallagher, R. H., et al, "Finite Elements in Fluids", vol. 6, John Wiley & Sons, 1985.
- 11 Gear, C.W., Leimkuhler, B., Gupta, G.K., "Automatic integration of Lagrange equations with constraints.", Journal of Computational and Applied Mathematics, 12&13, (77-90), 1970.
- 12 Gear, C.W., Petzold, L.R., "Singular implicit ordinary differential equations and constraints", Proc. Inter-american Workshop on Numerical Methods, June 14-18, 1982,

- Caracus, Venezuela, Lecture Notes in Math. 1005 (Springer, Berlin), (120-127), 1983.
- 13 Gear, C.W., Petzold, L.R., "ODE methods for the solution of differential/algebraic systems", Siam J. Numerical Analysis, vol.21, No.4, (716-728), 1984.
 - 14 Gilbrech, D. A., "Fluid Mechanics", Wadsworth Engineering Science Series, Wadsworth Publishing Company, Inc., (462-499) 1972.
 - 15 Hammersley, J. M., Handscomb, D. C., "Monte Carlo Methods", Mathuen & Co Ltd, 1965.
 - 16 Hess, L. L., Smith, A. M. O., "Calculation of Potential Flow about Arbitrary Bodies", Progress in Aeronautical Sciences, vol. 8, Pergamon Press, (1-38), 1967.
 - 17 Hocking, G., Mustoe, G.G.W., Williams, J.R., "Validation of the CICE code for ice ride-up and ice ridge cone interaction", Proc. Confer. Arctic'85, (962-970), 1985.
 - 18 Jenkins, J.T., Cowin, S.C., "Theories for flow of granular materials", Symp. Mechanics Applied to the Transport of Bulk Materials, American Society of Mechanical Engineers, Buffalo, N.Y., (79-89), 1979.
 - 19 Jenkins, J. T., Savage, S. B., "A theory of rapid flow of identical, smooth, nearly elastic, spherical particles", Journal of Fluid Mechanics, **130**, (187-202), 1983.
 - 20 Jenkins, J.T., Satake, M., "Mechanics of Granular Materials: New Models and Constitutive Relations", Elsevier Science Publ., Amsterdam, 1983.
 - 21 Kalos, M., H., Whitlock, P., A., "Monte Carlo Methods", vol. 8, John Wiley & Sons, 1968.
 - 22 Kanatani, K. I., "A micropolar continuum theory for the flow of granular materials", International Journal of Engineering Sci. vol. 17, (419-432), 1979.
 - 23 Lapidus, L., Pinder, G. F., "Numerical Solutions of Partial Differential Equations in Science and Engineering", John Wiley & Sons, 1982.
 - 24 Lun, C.K.K., Savage, S.B., Jeffrey, D.J., Chepurniy, N., "Kinetic theories for granular flow: inelastic particles in Couette flow and slightly inelastic particles in a general flow field", Journal of Fluid Mechanics, vol. 140, (223-256), 1984.

- 25 Malvern, Lawrence E., "Introduction to the Mechanics of a Continuous Medium", Prentice-Hall, Inc., 1969.
- 26 Ogava, S., Umemura, A., Oshima, M., "On equations of fully fluidized granular materials", Journal of Applied Mathematics and Physics (ZAMP), vol. 31, (483-493), 1980.
- 27 Plapp, J. E., "Engineering Fluid Mechanics", Prentice-Hall, Inc., 1974.
- 28 Roberson, R. E., Schwertassek, R., "Dynamics of Multibody Systems", Springer-Verlag Berlin Heidelberg, 1988.
- 29 Rowe, R.K., Davis, E.H., "The behaviour of anchor plates in sand", Geotechnique, vol. 32, (25-41), 1987.
- 30 Shen, H. H., Ackermann, N. L., "Constitutive relationship for fluid-solid mixtures", Journal of the Engineering Mechanics Division, American Society of Civil Engineers, **108**, (748-763), 1982.
- 31 Springer, W.A.J., "A Simulation of the Motion of Clusters of Rigid Bodies Around Obstruction", M.Sc. thesis, Department of Mechanical Engineering, University of Calgary, 1989.
- 32 Tatinclaux, J.C., Lee, C.L., "Initiation of ice jams - a laboratory study", Canadian Journal of Civil Engineering, vol. 5, (202-212), 1978.
- 33 Ting, J.M., Corkum, B.T., "Discrete element model in Geotechnical Engineering", 3rd Int'l Conf. Computing in Civil Engineering, 1988.
- 34 Tritton, D. J., "Physical Fluid Dynamics", Van Nostrand Reinhold (UK) Co. Ltd., England, 1977.
- 35 Vinogradov, O.G., "Simulation methodology for a flow of interactive ice floes around an obstacle", International Journal of Modelling & Simulation, vol. 7, no. 1, (28-31), 1985.
- 36 Vinogradov, O.G., "Simulation methodology of vessel-ice floes interaction problem", Proc. 5th Int'l Offshore Mechanics and Arctic Eng. (OMAE) Symp., vol 4, (601-606), 1986.
- 37 Vinogradov, O.G., Springer, W.A.J., "Algorithm for simulating flow of interacting rigid bodies", Proc. ASM '88, (134-138), 1988.

- 38 Vinogradov, O.G., Wierzba, P., "Simulation of motion of clusters of ice floes", Proc. ASM '86, (408-411), 1986.
- 39 Wells, D.A., "Theory and Problems of Lagrangian Dynamics", Schaums Outline Series, McGraw Hill Book Company, 1967.

A P P E N D I X

APPENDIX A
DERIVATION OF GENERALIZED INERTIA FORCES L_i

The kinetic energy of the system of N particles, or material points is given by:

$$T = \frac{1}{2} \sum_{j=1}^N (m_j \vec{v}_j \cdot \vec{v}_j) \quad (\text{A.1})$$

where $\vec{v}_j$ is a velocity vector of a particle given by:

$$\vec{v}_j = \frac{d\vec{r}_j}{dt} = \frac{\partial \vec{r}_j}{\partial t} + \sum_{k=1}^n \frac{\partial \vec{r}_j}{\partial q_k} \dot{q}_k \quad (\text{A.2})$$

Let us now recall that the generalized inertia forces are given by:

$$L_i = \sum_{j=1}^N m_j \vec{a}_j \cdot \frac{\partial \vec{r}_j}{\partial q_i} = \sum_{j=1}^N m_j \frac{d\vec{v}_j}{dt} \cdot \frac{\partial \vec{r}_j}{\partial q_i} \quad (\text{A.3})$$

and

$$L_i = \frac{d}{dt} \sum_{j=1}^N m_j \vec{v}_j \cdot \frac{\partial \vec{r}_j}{\partial q_i} - \sum_{j=1}^N m_j \vec{v}_j \cdot \frac{d}{dt} \frac{\partial \vec{r}_j}{\partial q_i} \quad (\text{A.4})$$

Let us now consider some of the terms in Equation (A.4).

Using Equation (A.2), we can write:

$$\frac{d}{dt} \frac{\partial \vec{r}_j}{\partial q_i} = \frac{\partial^2 \vec{r}_j}{\partial q_i \partial t} + \sum_{k=1}^n \frac{\partial^2 \vec{r}_j}{\partial q_i \partial q_k} \dot{q}_k = \frac{\partial \vec{v}_j}{\partial q_i} \quad (\text{A.5})$$

Also, from Equation (A.2) it can be seen that the velocity vector $\vec{v}_j$ is a linear function in the derivatives of generalized coordinates $\dot{q}_i$. It follows from here that:

$$\frac{\partial \vec{r}_j}{\partial q_i} = \frac{\partial \vec{v}_j}{\partial \dot{q}_i} , \quad (i = 1, n ; j = 1, N) \quad (\text{A.6})$$

Substituting Equation (A.5) and Equation (A.6) into Equation (A.4) we obtain,

$$L_i = \frac{d}{dt} \sum_{j=1}^N m_j \vec{v}_j \cdot \frac{\partial \vec{v}_j}{\partial \dot{q}_i} - \sum_{j=1}^N m_j \vec{v}_j \cdot \frac{\partial \vec{v}_j}{\partial q_i} \quad (\text{A.7})$$

which can be written as

$$L_i = \frac{d}{dt} \left[\frac{\partial}{\partial \dot{q}_i} \sum_{j=1}^N \frac{1}{2} m_j \vec{v}_j \cdot \vec{v}_j \right] - \frac{\partial}{\partial q_i} \sum_{j=1}^N \frac{1}{2} m_j \vec{v}_j \cdot \vec{v}_j \quad (\text{A.8})$$

And finally, by Equation (A.1) we have

$$L_i = \frac{d}{dt} \frac{\partial T}{\partial \dot{q}_i} - \frac{\partial T}{\partial q_i} \quad (\text{A.9})$$

APPENDIX B CALCULATION OF KINETIC ENERGY FOR RIGID BODIES

A motion of any rigid body can be considered as a combination of a translational motion and a rotation with respect to some point within the body. Figure B.1 shows a typical situation. We define $\vec{r}_0$ as a positional vector of the point of rotation and $\vec{r}_b$ as a positional vector from this point to some other point in the body.

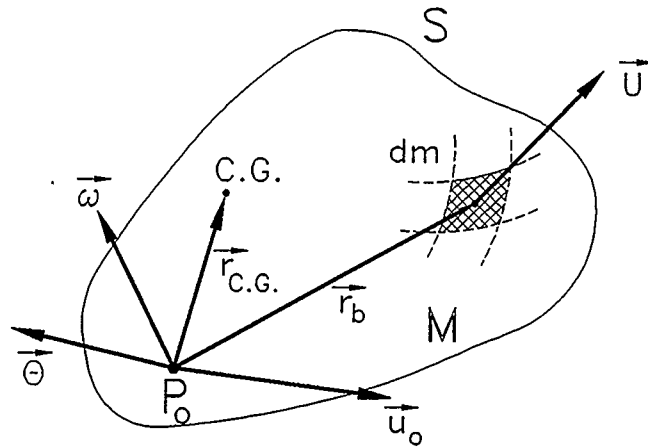


Figure B.1. Diagram for calculating T .

The positional vector to any point in the body can be defined as

$$\vec{r} = \vec{r}_0 + \vec{r}_b \quad (\text{B.1})$$

The velocity of any point of the body can be calculated as:

$$\vec{u} = \frac{d\vec{r}}{dt} = \frac{d\vec{r}_0}{dt} + \frac{d\vec{r}_b}{dt} = \vec{u}_0 + \frac{d\vec{r}_b}{dt} \quad (\text{B.2})$$

Term $\vec{u}_0$ represents the velocity of the chosen point "of rotation" P_0 , and represents the translational part of the motion, while term $(d\vec{r}_b/dt)$ represents the rotation. Since the positional vector $\vec{r}_b$ spans two given points of the rigid body, it follows that its length does not change. Its orientation, however, does, and hence, $(d\vec{r}_b/dt) = \vec{\omega} \times \vec{r}_b$. We can write Equation (B.2) as

$$\vec{u} = \vec{u}_0 + \vec{\omega} \times \vec{r}_b \quad (\text{B.3})$$

The kinetic energy of the body is given by

$$\begin{aligned} T &= \frac{1}{2} \int_M \vec{u} \cdot \vec{u} \, dm = \frac{1}{2} \int_M (\vec{u}_0 + \vec{\omega} \times \vec{r}_b) \cdot (\vec{u}_0 + \vec{\omega} \times \vec{r}_b) \, dm = \\ &= \frac{1}{2} \int_M \vec{u}_0^2 \, dm + \int_M (\vec{u}_0 \cdot \vec{\omega} \times \vec{r}_b) \, dm + \frac{1}{2} \int_M (\vec{\omega} \times \vec{r}_b)^2 \, dm \end{aligned} \quad (\text{B.4})$$

For the simplicity of expressions we will use a square to denote a dot product of two identical vectors, i.e. $\vec{a}^2 = \vec{a} \cdot \vec{a}$. In Equation (B.4) $\vec{u}_0$ and $\vec{\omega}$ do not depend on the position of a point within the rigid body, and hence can be taken outside the integration. We also, notice that

$$\int_M \vec{r}_b \, dm = M \vec{r}_{c.g.}, \quad \int_M (\vec{\omega} \times \vec{r}_b)^2 \, dm = I_\omega \vec{\omega}^2$$

where $\vec{r}_{c.g.}$ is the positional vector from point P_0 to the

body's center of gravity, and I_ω is the body's moment of inertia with respect to the axis along $\vec{\omega}$. We can now write Equation (B.4) as

$$T = \frac{1}{2} M \vec{u}_0^2 + M \vec{u}_0 \cdot \vec{u}_{C.G.} + \frac{1}{2} I_\omega \vec{\omega}^2 \quad (\text{B.5})$$

where $\vec{u}_{C.G.} = (d\vec{r}_{C.G.}/dt) = \vec{\omega} \times \vec{r}_{C.G.}$ is the velocity of the body's center of gravity relative to the point about which the rotation was considered.

Equation (B.5) represents the kinetic energy of a rigid body of mass M and having a moment of inertia I_ω with respect to the instantaneous axis of rotation $\vec{\omega}$. Alternately, Equation (B.5) could be written as

$$T = \frac{1}{2} M \left(\frac{d\vec{r}_0}{dt} \right)^2 + M \frac{d\vec{r}_0}{dt} \cdot \frac{d\vec{r}_{C.G.}}{dt} + \frac{1}{2} I_\omega \left(\frac{d\vec{\theta}}{dt} \right)^2 \quad (\text{B.6})$$

where $\vec{\theta}$ is a vector of rotation indicating the body's orientation in the inertial frame of reference.

APPENDIX C

TYPES OF PSEUDO-RANDOM NUMBER GENERATORS

The earliest attempt in generating a PRN sequence is due to von Newman. This is known as 'midsquare' method in which a number is generated by squaring its predecessor and taking the middle digits and is given by the following formula:

$$x_{n+1} = \left[\frac{x_n^2}{b^\alpha} - \frac{x_n^2}{b^{3\alpha}} \right] \cdot b^{2\alpha} \quad (\text{C.1})$$

This generator was, however, found unsatisfactory as it becomes a cyclic one, often with a very short period [13].

Most of the modern general use PRN generators are based on linear recurrence formula, [12],[13]. Here,

$$x_{n+1} = a_0 x_n + a_1 x_{n-1} + \dots + a_j x_{n-j} + b \pmod{P} \quad (\text{C.2})$$

where a_j , b , P , are constants. The generator is initiated by providing the first $j+1$ numbers $x_0, x_1, \dots, x_j$. Every PRN generator is characterised by a certain period or a number of times that the generator can be used before the sequence starts to repeat itself. For the generator given by Equation

(C.2) the maximum period that can be attained is: $\tau_{\max} = P^{(j+1)}$. The length of the period depends on the choice of the generator constants. In addition to achieving a maximum period one is also concerned with generating a sequence of numbers which in a statistical sense is indistinguishable from truly random sequences. Whereas the period τ can be predicted from the choice of the constants, the statistical properties of the PRN sequence are very much a matter of trial and error.

The generators of Equation (C.2) give numbers between 0 and P , i.e. $0 \leq x_i \leq P$. These can be mapped on to $(0,1)$ interval according to:

$$\xi_i = \frac{x_i}{P} \quad (\text{C.3})$$

Two important special cases of the generator of Equation (C.2) that are frequently used are the Multiplicative Congruential Generator and the Mixed Congruential Generator.

Multiplicative Congruential Generator is obtained from Equation (C.2) for $b = 0$, $a_j = 0$, ($j \geq 1$) :

$$x_{n+1} = \lambda x_n \pmod{P} \quad (\text{C.4})$$

For this type of generator the maximum period $\tau_{\max}$ is always less than P . Since P is an integer it can always be represented as a unique product of prime numbers. It follows from here that:

$$P = 2^\alpha q_1^{\beta_1} q_2^{\beta_2} \dots q_n^{\beta_n}$$

where $q_1, q_2, \dots, q_n$ are distinct primes. The maximum period will be equal to the lowest common multiple (LCM) of the individual periods. We write:

$$\tau_{\max}(P) = LCM\{\tau(2^\alpha), \tau(q_1^{\beta_1}), \dots, \tau(q_n^{\beta_n})\} \quad (C.5)$$

where the individual periods $\tau(q_i^{\beta_i})$ are computed according to:

$$\begin{aligned} \tau(q_i^{\beta_i}) &= q_i^{\beta_i-1} (q_i - 1) \\ \text{and} \quad \tau(2^\alpha) &= \begin{cases} 1 & (\alpha = 0, 1) \\ 2 & (\alpha = 2) \\ 2^{\alpha-2} & (\alpha > 2) \end{cases} \end{aligned} \quad (C.6)$$

The maximum period $\tau_{\max}$ as given by Equation (C.5) can always be achieved provided that parameter λ satisfies the following conditions:

$$\begin{aligned} \lambda^n &\neq 1 \pmod{q_j^{\beta_j}} & (0 < n < \tau(q_j^{\beta_j})) \\ \lambda &\equiv 1 \pmod{2} & (\alpha = 1) \\ &\equiv 3 \pmod{4} & (\alpha = 2) \\ &\equiv 3, 5 \pmod{8} & (\alpha > 2) \end{aligned} \quad (C.7)$$

and that the initial (starting) number x_0 is prime relative to P .

Mixed Congruential Generator is obtained from Equation (C.4) when parameter b in Equation (C.2) is no set to zero. Hence, the generator is governed by:

$$x_{n+1} = \lambda x_n + b \pmod{P} \quad (\text{C.8})$$

Here, unlike in the case of multiplicative congruential generator, a full period P can always be achieved, i.e. $\tau_{\max} = P$. It appears [13] that the value of increment b has little effect on the behaviour of the generator and the maximum period can be attained as long as b and P have no common divisor. The statistical properties of the generator are mostly governed by parameter λ . In order to achieve a full period P , λ must satisfy the following conditions:

- 1) $\lambda \equiv 1 \pmod{q}$ for every prime factor q of P ,
- 2) $\lambda \equiv 1 \pmod{4}$ if P is a multiple of 4,

and x_0 must be prime with respect to P .

In most applications P is chosen to be 2^β for binary computers or 10^β for decimal computers. **Modulo** function is then equivalent to retaining only β significant digits from the expressions on the LHS's of equations (C.2) and (C.8). If β is taken as the word length of the computer, then the unwanted digits are automatically disregarded when the computer does integer arithmetics.

APPENDIX D

RIGID BODY GENERATION COMPUTER ROUTINE DESCRIPTION

A block diagram of a computer routine generating the rigid bodies and positioning them on the generation boundary is shown in Fig(D.1).

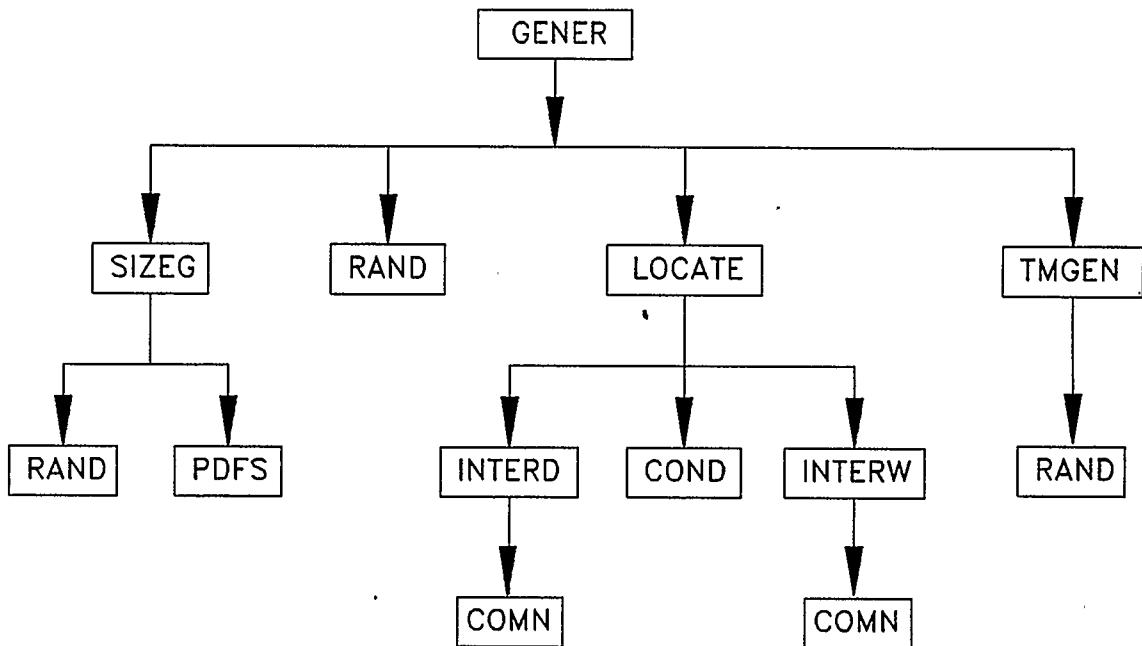


Figure D.1. Block diagram for the generation routine.

The subroutine and function subprograms that comprise the computer routine are:

GENER

This is the main subroutine. It sets up the generation

boundary and Poissonian interval for time increment generation, and performs other organisational tasks. It also generates the location of the rigid body along the generation boundary. This subprogram receives the necessary input from the main program, executes all other subroutines, and returns the output back to the main program. It is called every time a rigid body is to be generated.

SIZEG

This subroutine is responsible for generating the size of a rigid body according to the methodology described in Section 4.4.1. It uses two function subprograms: RAND which supplies it with pairs of random numbers, and PDFS which contains the assumed probability density function for the size distribution.

LOCATE

This subroutine locates the rigid bodies on the generation boundary i.e, determines the coordinates of the center of the disk representing the generated rigid body. If for the position supplied by GENER there is no overlap then it positions the rigid body at that point. If, however, the would-be overlap is detected it establishes all the discrete overlap intervals (Section 4.4.4) and adjusts the position accordingly. It uses

INTERD, INTERW, COND, and COMN subroutines for the task of identifying the overlap intervals.

INTERD

This subroutine establishes the overlap interval between a disk and a generation boundary line segment. It uses methodology described in Section 4.4.4, Equations (4.45 - 4.48), and subroutine COMN for this purpose.

COMN

This subroutine determines the intersection of two given intervals on the real axis, i.e. it finds x_1, x_2 such that

$$(x_1, x_2) = (a_1, a_2) \cap (b_1, b_2)$$

where $(a_1, a_2), (b_1, b_2)$ are the two intervals. If the intersection is an empty set then it returns an appropriate message.

INTERW

This subroutine identifies the overlap intervals on the generation boundary which are caused by the solid boundary line segments, (Section 4.4.4). It uses COMN subroutine similarly to INTERD.

COND

This subroutine condenses an arbitrary set of subintervals in the given interval into a discrete set.

TMGEN

This subroutine determines the random time increment to the next rigid body generation. It samples a Poissonian distribution according to the methodology described in Section 4.4.3. It uses RAND function subprogram to provide it with random numbers on (0,1).

RAND

This function subprogram comprises a pseudo-random number generator. It generates a PRN sequence according to the governing Formula (4.5) in Section 4.2. This routine reseeds itself.

PDFS

This function subprogram contains the probability density function for the size distribution. The PDF can be both continuous or discrete.

APPENDIX E
DERIVATION OF THE FLUID FLOW EQUATIONS

Conservation of Mass

Let us consider an arbitrary volume V fixed in space and entirely within the fluid enclosed by a control surface S , as shown in Figure E.1.

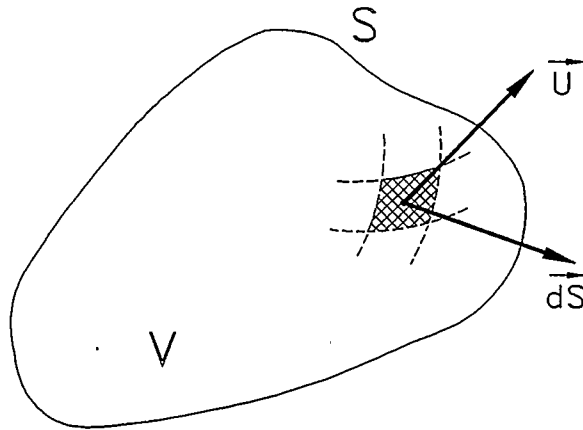


Figure E.1 A schematic diagram for the formulation of the continuity equation.

Conservation of mass dictates that the net mass flux through surface S be equal to the rate of the accumulation of mass inside volume V with a negative sign. This condition can be written as

$$\frac{\partial}{\partial t} \int_V \rho dV = - \int_S \rho \vec{U} \cdot d\vec{S} \quad (\text{E.1})$$

where: ρ is the fluid density and $\vec{u}$ is the velocity vector. Applying the divergence theorem to the surface integral, equation (E.1) can be rewritten as:

$$\int_V \frac{\partial \rho}{\partial t} dV + \int_V \text{div}(\rho \vec{u}) dV = 0 \quad (\text{E.2})$$

Since Equation (E.2) is satisfied for any arbitrary volume, it follows that,

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{u}) = 0 \quad (\text{E.3})$$

The above result is known as the continuity equation.

Conservation of linear momentum

In Equation (E.1) a partial differentiation with respect to time, of an integral over some spatial volume V represents a local rate of change of the total quantity (mass, in Equation (E.1)) inside this volume. In setting up the equation for the conservation of momentum we need to determine the rate of change of a quantity (momentum, here) while following the mass system that instantaneously occupies a given spatial volume. In this case not only does the integrand change with time, but so does the volume over which the integral is taken. Such a rate of change is denoted by (d/dt) and is called a material time derivative. A material

time derivative operator is given by the following relation:

$$\frac{d}{dt} = \frac{\partial}{\partial t} + \vec{u} \cdot \nabla \quad (\text{E.4})$$

The operator can be applied to a scalar, a vector, or a tensor. The first term in equation (E.4) is referred to as the local rate of change at a given point in space, whereas the second term describes the convective rate of change in the neighbourhood of a fluid particle as it moves to a different point.

We will now compute a rate of change of the total amount of the quantity (which could be a mass, momentum, energy, etc.) that is carried by a mass system which instantaneously occupies a certain spatial volume V . If $\mathcal{E}$ denotes the quantity per unit mass then the rate of change of the total amount of this quantity is given by Reynolds transport theorem.

$$\frac{d}{dt} \int_V \mathcal{E} \rho \, dV = \int_V \frac{\partial}{\partial t} (\mathcal{E} \rho) \, dV + \int_S \mathcal{E} \rho \vec{u} \cdot d\vec{S} \quad (\text{E.5})$$

Utilizing the divergence theorem, the continuity equation (E.3), and Equation (E.4), the above relationship can be simplified:

$$\frac{d}{dt} \int_V \mathcal{E} \rho \, dV = \int_V \rho \frac{d\mathcal{E}}{dt} \, dV \quad (\text{E.6})$$

Quantity $\mathcal{E}$ can be a scalar, a vector, or a tensor. In a specific case if this quantity represents, for example, a velocity vector, i.e. the momentum per unit mass, then:

$$\frac{d}{dt} \int_V \rho \vec{u} dV = \int_V \rho \frac{d\vec{u}}{dt} dV = \int_V \rho \left(\frac{\partial \vec{u}}{\partial t} + \vec{u} \cdot \nabla \vec{u} \right) dV \quad (\text{E.7})$$

would represent a rate of change of the total momentum possessed by a given mass system that instantaneously occupies a spatial volume V . We can now formulate the momentum equation for a moving fluid. Newton's Second Law of motion. states that the rate of change of the momentum experienced by a fluid particle is equal to the net force acting on it. Let V represent a certain spatial volume enclosed by a control surface S and entirely within the fluid, as shown in Figure E.2).

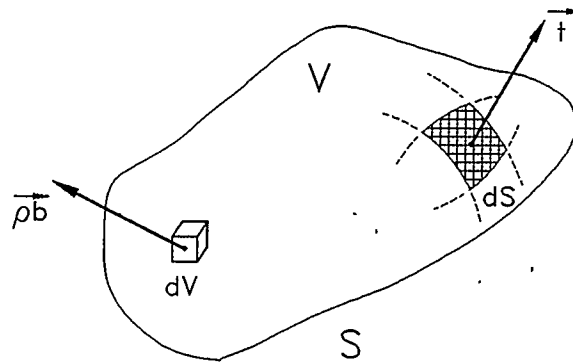


Figure E.2 A schematic diagram of momentum balance.

Providing that Newton's Third Law of action and reaction

holds, the rate of change of total momentum of the mass system that instantaneously occupies the given volume V is equal to the total external force acting on this mass system. Using the result of Equation (E.7) this can be written as

$$\int_V \rho \frac{d\vec{u}}{dt} dV = \int_V \rho \vec{b} dV + \int_S \vec{t} dS \quad (\text{E.8})$$

where $\vec{t}$ is the vector of external force per unit area acting on the mass system and $\vec{b}$ is the vector of a body force per unit mass. The external forces acting on the mass system are equilibrated by the internal stresses according to

$$\vec{t} = \hat{n} \cdot \mathbf{T} \quad (\text{E.9})$$

where $\mathbf{T}$ is the symmetric two-dimensional stress tensor and $\hat{n}$ is the unit normal vector of the surface. Substituting Equation (E.9) and using the divergence theorem to transform the surface integral, Equation (E.8) may be written as

$$\int_V \rho \frac{d\vec{u}}{dt} dV = \int_V (\nabla \cdot \mathbf{T} + \rho \vec{b}) dV \quad (\text{E.10})$$

Since Equation (E.10) is satisfied for any arbitrary volume V , it follows that

$$\rho \frac{d\vec{u}}{dt} = \nabla \cdot \mathbf{T} + \rho \vec{b} \quad (\text{E.11})$$

The above result is known as the Cauchy's equations of motion

or Cauchy's first law of motion. The stress tensor $\mathbf{T}$ can be related to the rate of deformation tensor $\mathbf{D}$ by the following Navier-Poisson law of a Newtonian fluid:

$$\mathbf{T} = -p\mathbf{I} + \lambda \nabla \cdot \vec{u} \mathbf{I} + 2\mu \mathbf{D} \quad (\text{E.12})$$

where $\mathbf{I}$ is the identity tensor and p is the fluid pressure at a given point. The rate of deformation and the identity tensors are given by the following relation:

$$D_{ij} = \frac{1}{2} \left[\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right], \quad I_{ij} = \delta_{ij} \quad (\text{E.13})$$

Combining Equations (E.11), (E.12), and (E.13) the momentum equation for a Newtonian fluid may be written as

$$\rho \frac{d\vec{u}}{dt} = -\nabla p + (\lambda + \mu) \nabla (\nabla \cdot \vec{u}) + \mu \nabla^2 \vec{u} + \rho \vec{b} \quad (\text{E.14})$$

Equation (E.14) is known as the generalized Navier-Stokes equation for a Newtonian Fluid. In the equation μ is the coefficient of viscosity and λ is the second viscosity coefficient.

APPENDIX F

MOMENTUM EQUATION FOR POTENTIAL FLOW

The general form of the momentum equation as derived in Appendix E, Equation (E.14), is

$$\rho \frac{d\vec{u}}{dt} = -\nabla p + (\lambda + \mu) \nabla (\nabla \cdot \vec{u}) + \mu \nabla^2 \vec{u} + \rho \vec{b}$$

Since the flow is potential then the velocity vector is:

$$\vec{u} = \nabla \phi \quad (\text{F.1a})$$

This also means that the flow is incompressible and irrotational. These two conditions can be written as

$$\nabla \cdot \vec{u} = 0 \quad (\text{F.1b})$$

$$\vec{\omega} = \nabla \times \vec{u} = 0 \quad (\text{F.1c})$$

By the incompressibility condition (F.1b) the term involving λ drops out from the momentum equation, and it can be written as:

$$\rho \frac{d\vec{u}}{dt} = -\nabla p + \mu \nabla^2 \vec{u} + \rho \vec{b} \quad (\text{F.2})$$

Let us now consider a following vector identity:

$$\nabla^2 \vec{u} = \nabla(\nabla \cdot \vec{u}) - \nabla \times (\nabla \times \vec{u}) \quad (\text{F.3})$$

The first term on the left hand side of Equation (F.3) vanishes by the incompressibility condition (F.1b). The second term represents a curl of the vorticity vector. Thus for an incompressible fluid:

$$\nabla^2 \vec{u} = \nabla \times \vec{\omega} \quad (\text{F.4})$$

Substituting the result of Equation (F.4) into Equation (F.2) the momentum equation becomes:

$$\rho \frac{d\vec{u}}{dt} = -\nabla p + \mu \nabla \times \vec{\omega} + \rho \vec{b} \quad (\text{F.5})$$

The above result suggests an analogy between the viscosity and the vorticity. The viscous term ($\mu \nabla \times \vec{\omega}$) vanishes from the momentum equation (F.5) for either an irrotational flow ($\vec{\omega} = 0$), or a flow of a perfect (inviscid) fluid ($\mu = 0$). Hence, with the use of Equation (E.4) we may write the momentum equation for an incompressible and irrotational fluid as:

$$\rho \frac{\partial \vec{u}}{\partial t} + \rho (\vec{u} \cdot \nabla) \vec{u} = -\nabla p + \rho \vec{b} \quad (\text{F.6})$$

Consider another vector identity:

$$(\vec{u} \cdot \nabla) \vec{u} \equiv \frac{1}{2} \nabla (\vec{u} \cdot \vec{u}) - \vec{u} \times (\nabla \times \vec{u}) \quad (\text{F.7})$$

Since the flow is irrotational the second term on the right hand side of identity (F.7) vanishes and the momentum equation takes on the following form:

$$\rho \frac{\partial \vec{u}}{\partial t} + \frac{1}{2} \rho \nabla (\vec{u} \cdot \vec{u}) = -\nabla p + \rho \vec{b} \quad (\text{F.8})$$

Rearranging the terms and using Equation (F.1a)) an alternate form of the momentum equation for an irrotational and incompressible flow is obtained.

$$\nabla \left(\frac{\partial \phi}{\partial t} + \frac{1}{2} u^2 + \frac{p}{\rho} \right) = \vec{b} \quad (\text{F.9})$$

Equation (F.9) represents the final form of the momentum equation for a potential flow.