Chapter 2

Literature Study/
Library Study
Chapter 2: Literature Study/Library Study

Even though on related topics: grid floors and plated structures, considerable contributions have been made, the form of Grid-Nodal-Matrix-Based stiffeners (implanted herein) has not been found reported anywhere (to the best of my information sources). So search for related literature, with an eye on structural innovations, identified the material used here as spadework for this thesis which, it may not be out of place, to spell out as hereunder:

2.1 Mies' Concept of Universal Space: Significance
2.2 Structure and Form: Significations
2.3 Significance of Model Analysis for Design of Buildings
2.4 Engineering Section: General
2.4.1 Load Transfer in Two-Directions: Grid-Action Generation Study.
2.4.2 Finite Element Method: Introduction
2.4.2.1 Finite Element Method: An Appraisal
2.4.2.1.1 Applications of Finite Element Method
2.4.2.1.2 Advantages
2.4.2.1.3 Limitations
2.4.2.1.4 Conclusions
2.4.2.2 Convergence Requirements
2.4.2.3 STAAD-III for Windows/Windows NT: Literature Reviewed
2.4.2.4 The ANSYS Program: Literature Reviewed.
2.1 Mies' Concept of Universal Space: Significance

This new principle propounded by him in year about 1940 — provided a major break from earlier trends of (singular) concern for functional needs in plan(s)/design(s) development strategy. Contrary to Sullivan's idea, form follows function, Mies emphasised: "that while form cannot change, function does. And we do not let the function dictate the plan. Instead let us make room enough for any function." In not establishing a predetermined and contained allocation of space to accommodate the functional necessities of a building, but by providing a rational space capable of continual modification and thereby designs to counteract obsolescence.

This idea reflected again the present polarity of approach to design. Mies believed strongly — that with the vast increasing scale of our society, and with (its) advancing technological and scientific forces/changes and demands— it tends to negate the specific solution, and instead these forces demand a universal solution: a flexible response to the need for space at a vast scale; a more anonymous background giving maximum opportunity for individual liberty.

To Wright this was an alien generalization— as his buildings demonstrate. To him, the universal approach was an abstraction synonymous with the anonymous man; and he saw the personal space as proper response to individual needs in a fixed time and place.

Applications (Examples)

2.1.1 Truss Construction with Suspended Roofs Systems

With the aim of providing complete flexibility of arrangement in the interior, a vast hall-like structure has been created entirely free of internal columns. The roof is suspended from a system of trusses, which are supported upon outside columns, thus leaving the interior entirely unobstructed. This building simply contains a large space enclosed by the flat suspended roof and the vertical outer skin of glass. It, therefore, achieves the ultimate in unity of spatial, aesthetic and technological organisation.

This truss construction system for large halls can serve a variety of function. It has been employed for a school building (for architecture) and captured in a theatre project.

(1) Crown Hall, Architecture and Design Building
IIT, 1952-56

This glass and steel hall, which is free from inside supports, provides a work centre for the students and staff of the faculty of architecture and town
planning (working together in such close proximity makes for an easy relationship between students and benefits their studies). Students work is regularly on display in the central exhibition area of the hall.

The roof of the building measures 120 x 220 feet. It is suspended from four welded plate-girders which span the entire width of the roof at 60-foot centres. At the ends, the roof slab projects 20 feet beyond the outer truss. A glass-and-steel wall encloses the building. The lower panels of glass are sand-blasted; the upper windows and those at the entrance are of transparent glass, fitted on the inside above the sand-blasted glass, are Venetian blinds which are kept in a down position to ensure an even distribution of light.

A monumental flight of travertine steps leads, by way of a platform terrace, to the main entrance level, which is six feet above the ground. The white acoustic ceiling is 18 feet above the terrazzo floor. The free standing partitions are of oak.

2.1.2 Glass House with a Steel Frame

The columns are located outside so as to give infinite flexibility in internal arrangement. The construction of the roof slab depends on whether the columns are arranged along the two long sides of the house or all around its perimeter.

The use of steel and glass gives an uninterrupted view of the natural surroundings and enables the interior space to be projected outside. A pedestal or terrace in front of the building fits the house into the environment with a sense of belonging.

These steel-and-glass structures are set like crystals amidst the luxuriance of nature. Only the art of omission reveals the true structure of a building and reduces it to elements of pure beauty and pure spirit.

(1) Farnsworth House Single-Room Glass House
Plano (Illinois), 1945-1950

This week-end house with an one room first floor stands on a flat meadow between tall leafy trees. The living side faces the Fox River, which forms the southern limit of the site. The roof and floor slab are raised above the ground by eight outside steel columns and enclosed by a glass skin. An inner core of natural primavera wood, containing the service installations, partitions off the kitchen, sleeping and living areas.
A raised terrace in front of the house forms a link with the lawn. The various levels are joined by two flights of steps. The steps, the terrace and the floor are faced with travertine slabs measuring 2x3 feet. All exposed steel elements are painted white. The transparent sheets of glass can be screened by curtains of natural-coloured shantung. The dimensions of the house are 77x29 feet. The pedestal measures 55x22 feet, and the interior is 9.5 feet high. The columns are placed 22 feet apart.

2.1.3 Hall Construction With Wide-Span Roofs

Since it is the structure that determines the essential character of a building by Mies van der Rohe, he can design monumental hall buildings, which are masterpieces of engineering, and yet display an exquisite sensibility. A wide-span roof supported only on outside columns yields a free-space in which the creative imagination can be given full play. The freedom provided by a static structure is the essence of Mies van der Rohe's work. This "static" structure is, as it were, the instrument on which his creative genius can play the "dynamics" variations of his designs.

Mies van der Rohe does not build palaces or heavy, massive fortresses. For a minimum of mass, his buildings yield a maximum in cultivated living: Living in a state of freedom.

(1) Convention Hall, Chicago

Project 1953-54

The Convention Hall was planned for a site near the centre of Chicago within easy reach of all transportation facilities. It would be a multi-purpose hall with seating accommodation for 50,000 people. A column-free interior would meet every demand for exhibitions, sports events and conventions.

The square roof consists of a steel framework unit, 30 feet height and 720 feet in span. It rests upon all four outer wall which take the form of trusses (60 feet in height) raised 20 feet above ground on six tapered concrete columns. The total height of the hall from floor to ceiling is 110 feet.

The glazed walls of the ground floor are set back 30 feet. Dark grey marble, aluminium or tinted glass fill the panels so that the structure of the hall is expressed clearly both inside and outside. Mies has always built so that "the construction is the building itself".
(2) Museum of Fine Arts, Houston  
Project, 1954-58  
The hall is built in the courtyard of the existing neoclassical art museum. Four steel girders 1.5 metres high and 24 metres long hold the symmetrical roof slab. The column-free interior is 9 metres high. Visible walls of pale grey brick link the old and the new buildings. The hall of the Museum of Fine Arts is a large hall similar in concept to the interior of the Crown Hall in Chicago.

(3) Bacardi Building, Santiago de Cuba  
Project 1957-59  
The building program called for a hall with a column-free interior which was to be built in concrete.

The roof measuring 54 x 54 metres is a rigid "egg crate" made up of intersecting concrete beams. At each of its outside edge, it is supported on two concrete columns. In section, the columns take the form of a cross with a hinge at the top to bear the weight of the roof. The hall is 7 metres in height and stands on a podium set in the sloping landscape. The interior space, set back and glazed on all sides, serves as a reception area for an administration department.

To the constantly repeated question as to why the National Gallery in Berlin is so similar in design to Bacardi in Santiago de Cuba, despite one being an administration centre and other a museum, Mies replies "I refuse to invent a new architecture every Monday morning. The Greeks needed hundreds of years to complete the Doric column, and it's all to do with completion".

(4) National Gallery, Berlin  
Project, 1962-68  
The square glass-enclosed hall for travelling art exhibitions stands on a broad terrace. Beneath the terrace is the gallery for the permanent collection, the administration and storage.

The site slopes down to its western end where the terrace forms a walled courtyard which provides daylight for the gallery rooms on the lower floor.

The roof of the hall is a flat, two-directional structure 1.8 metres deep. It consists of welded steel web-girders arranged at 3.6 metre centres in both directions and forming a square structural grid. The continuous plate is reinforced with ribs to prevent buckling. Eight steel columns, two on each side, support the roof. The roof structure measures 64.8 x 64.8 metres, and the interior of the hall is 8.4 metres high.
The glazed walls are set back 7.2 metres on all sides — so as to leave an arcade between the glass and the columns. The terrace and the floor of the hall are paved with granite slabs measuring 1.2 x 1.2 metres.

At the gallery level, the building is a reinforced concrete structure (column at intervals of 7.2 metres). The materials for the gallery rooms (four metres high) are plaster ceilings painted white; plywood panel walls, also painted white; and terrazzo for the floors.

The exhibition hall and museum spaces at the lower level will have an orderly arrangement of down lights for general illumination. In addition, the exhibition hall will be given a number of spot lights which can be inserted in the ceiling wherever needed. A specially designed wall washer system will light the picture walls.

The problem of arranging works of art in spaces has always interested Mies van der Rohe. Collages of earlier buildings, for example a Villa (1938) or the Museum for a Small City (1942), show how sculpture, painting and space can be integrated and indicate what the character of this museum will be.

(5) **Concert Hall**
Project, 1964

As a rule, a well-proportioned space always has good acoustics. So say major conductors, and the infinite flexibility in interior arrangements can be easily realized by Mies, thanks to a freedom from interior columns.

In this hall architecture, with its wide-span roof resting on external columns, a concert hall can be created with freely arranged wooden walls enclosing the space like the sides of a sound box. At the same time, the built-in elements are independent of the columns and the roof. Mies explained this idea in 1964 with the aid of a freehand drawing.

A fitting remark by Mies is: "I don't want to be interesting; I want to be good". With Mies, only the extraordinary applies - the wager. He does not know mediocrity. No fashions; no trends. The dictates of an ordered architecture mark his personality. The architecture of filling and supporting faithfully follows Mies' Idea that the physiognomy of a man may change, but his skeleton remains the same. And we may conclude this study by briefly reiterating the architectural principles taught by Mies at the IIT in Chicago:

1. The structure as an architectural factor; its possibilities and limitations;
2. Space as an architectural problem;
3. Proportion as a means of architectural expression;
4. The expression value of materials; and
5. Painting and sculpture in their relationship to architecture.
2.2 Structures and Form: Significations

(1) There can be no architecture without technology. The latter enables the realisation of concept as physical entity.

(2) Technology has always influenced building forms: and architects of every age have advanced their profession by the technical mastery of methods and materials. The Parthenon and the Gothic cathedrals are both essentially ultimate refinements of a particular technique using certain materials. Structural form does not imply something accidental and unique, but something typical and enduring.

(3) Structural form is not born of intuition alone. To discover and shape it—so that even a lay-user can understand it, technical knowledge is required.

(4) Structural form is not tied to any narrow trend in modern architecture. Its principles lie deeper; they are discernible in the architecture of the past and have now spread throughout the world, without being confined to any particular “school”.

(5) Structural form is an indispensable element of modern architecture.

(6) Structural form is a means of architectural expression typical of our times. It springs from a perfectly definite conception of the design process that recognizes natural order as the supreme law. As a means of expression it is comparable with language. Like the latter, it can swamp even the best ideas, leaving them misunderstood and neglected. At the same time, however, it can elevate an essentially modest statements to the level of a work of art with ultimate clarity and distinction.

(7) If this goal is to be achieved, if structural form is to become a purifying, ordering and constructive principle of modern architecture, we shall need to improve our understanding of its technical basis, and achieve a profound insight into its essential nature: of a governing phenomenon.

(8) In this situation it seems pertinent to broach the question of the genuineness of technological form in architecture and to attempt a logical and simple answer. The trouble is that the bases of architectural appreciation are never established by purely rational means.

(9) Since architecture necessarily includes a technological component, it must be possible to encompass at least this side of it within a rational system. In fact, modern architecture, being the architecture of a technological age, cannot afford to forgo a clarification of the technical problems of purification, simplification, genuineness, and (visual) pertinence.

(10) Whereas previously building technology was universally intelligible, today we are obliged to seek new ways of speaking and understanding the complicated engineering and scientific language—which have become an essential part of architecture.
Finally, we must make an effort to grasp the contemporary significance of technology; the importance of which was never questioned in earlier times, together with all the consequences of its pre-eminent position as a form-determining element, and bend them to the service of architecture, uncorrupted and without concession in the direction of formalism.

To accomplish this symbiosis we must be ready to bring architecture as an art right into the middle of a technological world without creating hostility between the two. We shall then see that technology, the nature of which is rooted in natural laws, will yield the "structural form", which constitutes the theme of this research work.
2.3 Significance of Model Analysis for Design of Buildings.

Building architecture differs from most other manufacturing industries. Its products are quite expensive, but generally only one article is made to each design. The cost of designing a building is between 5 to 10 percent of the total; and the cost of designing the structure, which is most amenable to model analysis, is of the order of 1 percent of the total cost of the building. It is often preferable to use an approximate calculation than to employ a precise analysis normally.

This is not true in the design of most other commodities. The cost of designing a plastic case for a radio receiver is many times the cost of a single receiver. This may not be important because great many receivers are made to a single design.

None of these arguments apply to the design of buildings. We cannot afford to break a complete full-size architectural structure as we can break a prototype plastic radio casing. The simplified geometry of the building may look a little different within the framework of modern technological means available.

Many of the techniques used in the model analysis of architectural structure are derived usually form the general manufacturing industry, the aircraft industry, and from the design of dams, and all may not be feasible—when applied to buildings.

It is perhaps in its least scientific aspects that the architectural model has so far been most successful. Most architects use a model to show the appearance of the building. These models are generally viewed from above, i.e. the point of view of an observer in an aeroplane, and consequently do not convey a correct impression. This defect could be readily corrected with a small periscope, but few people use one. The model is a means of explaining the architectural concept to the layman, it helps the architect to visualize the entire scheme, and it is rarely used for detailed architectural design, which is done of paper.

This aspect of model design has not been considered sufficiently in architectural science. Even where a structural model may not be the best means of obtaining the detailed dimensions of a structure, it can often help the architect and his client, and often also the structural consultant, to obtain better appreciation of the problem. The interrelation of the various forms of model analysis has not received sufficient attention. Model analysis could, therefore, be more economical if the various aspects were considered together initially, with due allowance for the scale-factors, appropriate to various forms of analysis.

2.3.1 Experimental versus Theory: Present Status

Theoretical and experimental analysis are essentially complementary forms of structural design.
In the late 18th century, the elastic theory made rapid progress, and iron was used more and more for structural design in the next few decades. Empirical rules gave way to the theoretical calculations, and structural systems were adapted to the existing simple theory.

Before the 20th century, the theory of structures was never used successfully for the design of complex masonry vaults and domes, which continued to be designed by empirical rules. The growing sophistication of structural theory was largely responsible for the increasing complexity of engineered mechanized form of structural design, as calculations became more laborious.

The apparatus devised by Beggs at Princeton University in 1922, the first successful method of model analysis, and other early examinations may be regarded as mechanical analogues of the elastic theory, rather than scale models of a part of the building. Their significance lay in the fact that gave the same answer as the mathematical solution; and it was incidental that the model dimensions were proportional to those of the structure.

In other fields of engineering the theory of dimensional analysis was well established in the early 20th century. Scale models were regularly used for hydraulic works, and in the 1920's wind-tunnels were commonly employed to test scale models of aeroplanes and aeroplane parts.

At the same time imaginative reinforced concrete structures of complex shape began to make their appearance in Southern Europe. Both Nervi and Torroja used small scale models in their designs. During the Second World War aircraft design made great progress, and structural designing of aeroplanes with the aid of scale models, instrumented with the new miniature elastic resistance strain gauges—was carried to a high degree of perfection.

In the late 1940's the same method was applied to architectural structures, and rapid design solution were obtained for problems hitherto solved only by exceedingly lengthy calculations. So, the Second World War had produced another powerful design tool, which took a little longer to be adapted in structural designs.

In the 1950's digital computer solutions for structural problems became available, and the scope of computer based designing is steadily increasing, since then.

### 2.3.2 Types of Structural models

Structural model studies may be conveniently divided into five main types, depending upon the principal objectives of investigation.

#### 2.3.2.1 Feasibility Study Models

In novel forms of construction it is economical to test the feasibility on a scale model. This enables the designers to assess the performance of the proposed method of construction, without erecting a full scale mock up.
The roof of the Sydney Opera House was built from precast concrete segments. The main shell arches are made up of a series of ribs, connected to a ridge beam at the apex. The center line of each rib is located on the great circle of a sphere of radius 246 ft, so that ribs have the same radius, but different lengths. Identical cast elements are used for all ribs and located at the same radial distance from the center of the great circles. Since two thousand ten segments had to be erected, the contractor designed a telescopic erection truss, capable of supporting all the ribs without the need for scaffolding.

To test the feasibility of the procedure the contractor built at fully workable (at 1:48) scale model of brass. The model contained electric motors which caused the truss to elongate the arch-length, whilst maintaining a constant radius— as was required in the prototype.

2.3.2.2 Demonstration Models

These are designed specifically to demonstrate one or more known aspects of behaviour. Under this heading there is a wide range of models varying both in size and in complexity. Classroom demonstration models are at their simplest. They should be portable, easy to operate and simple to comprehend. They can be made from inexpensive materials—even cardboard—and generally demonstrate their point of exaggeration. Because simplicity is required, they have little or no instrumentation.

And laboratory demonstration models are specifically designed for measurement, and thus are accurately made. As they are primarily intended for student use, they should be robust. They may have simple or complex instrumentation, depending on the type of model and the quantities to be measured. Their purpose is to enable students to measure some quantity: load, deformation or strain.

2.3.2.3 Behavioural Models

These are designed to demonstrate clearly some particular structural behaviour of principle. In order to do this well, it is often found advantageous to exaggerate the response of the model to various actions, by deliberately simulating extreme conditions or using model materials which magnify the structural responses.

Behavioural models are also used as a teaching aid to complement analytical instructions and as a medium for the preliminary semi-quantitative assessment of complex structures. Emphases on the visualisation of the desired behaviour is often accompanied by suppression of unwanted secondary effects, i.e. deliberately distorting similarity relationship. Therefore behaviour models are demanding much ingenuity of design, but a minimum of labour and expense.
The impact of behavioural models has influenced the philosophy of teaching structural principles to students of architecture and related disciplines. Teachers of architectural science have found this approach rewarding, and this aspect of model studies (since its introduction) has become an important part of the architectural science laboratory.

When the study forms part of the evolution of early structural concepts for a design, behaviour models are incalculable, e.g. during the early phase of the design of a stadium, as they help to resolve the potentiality and feasibility of alternative structural concepts.

For teaching purposes, behavioural models can be made of almost any material. For instance, moderately stiff paper is suitable for a convincing demonstration of the strength gain of slab structural by folding, and a block of foamed plastic can transform the rather abstract idea of the "middle third law" into a more lasting physical image for the movie in structural mechanics. Here wood, metal and plastic all have found application in model development, generally.

2.3.3.4 Confirmatory Models

As the name implies, provide corrective evidence for analytical models, and are usually conducted when circumstances are such as:

I The theoretical analysis is available, but the degree of uncertainty of the approximations made in the formulation and solution of the theory is not known, and

II An acceptable design method is available, but the magnitude and importance of the structure warrant an experimental verification.

Case(I) includes model studies on the acceptability of existing methods of theoretical analysis and the testing of improved mathematical formulations. It is therefore essential that the physical model is similar to the mathematical model, rather than to reality. This simplifies the problem as attention is focussed on the confirmation of an analytical solution, which, if successful, is modified and extended to cover the study for elastic theory of thin concrete shell structural, it is preferable to select near-elastic and homogenous material like Plexiglass for the model in spite of its dissimilarity to reinforced concrete.

Case (II) arises when the prototype is very expensive or the consequences of failure are disastrous, e.g. a tower structure of unprecedented proportion of a containment vessel in a nuclear energy generating establishment. A successful model correlation then provides a "second opinion" to theory, and raised the level of confidence in the subsequent structural design. For instance, the structural design of the 1,250 ft. (380m) high Empire State Building, built in 1933, was aided by a wide tunnel model study to check the magnitude and nature of the wind forces.
2.3.2.5 Design Models

These are the most expensive type. Architectural structures are intrinsically complex, and the separation of a structure into beams, columns and cladding is but an approximation born of expediency. While this simplification serves to render analytical design methods practicable, it is acceptable only as long as the results are consistent. If the approximation is excessive, either a grossly conservative or a dangerously unsafe structure results. If the structure is too difficult for an analytical solution, it is best to use a design mode.

Direct structural design on the basis of models is widely used for the design of aero-structures-demanding safety as well as minimum weight, and dams must be made safe in spite of the complex interaction between structure and supporting state. Applications to building structures have not been so spectacular, although they are gaining in significance as buildings become more daring. Direct design models are useful for highly indeterminate structure results. If the structure is too difficult for an analytical solution, it is best to use a design model.

Before making a model study the basis for design, three significant factors must be considered:

• availability of specialised facilities and personnel to ensure good reliability of test results obtained;
• acceptability of this method of design to the building authorities; and
• competitiveness in terms of time and money by comparison with an analytical method (if feasible).

In brief, currently, model analysis are conducted mainly by specialised university, government and commercial laboratories which have built up, over a number of years, the requisite facilities and expertise. The time and money required for an adequate model investigation depends on the accuracy required; but in complex problems, particularly, the model study may suggest improvements in the design – far in excess of the cost incurred.
2.4 Engineering Section: General

In this section, related techniques and advanced computer-methods, used as tools, for carrying out computer-aided analytical/theoretical study is reviewed, arranged in order, and presented herein for the users reference in subsequent sections.
2.4.1 Load transfer in two directions: Grid-Action Generation Study

The structural elements used commonly exhibit the property of transferring loads in one direction, mainly. A load set on a beam or cable is channeled to the supports along the beam axis or cable line; Similarly, an arch, a frame, and a continuous beam produce the same type of "one-directional load dispersal." These structures are labeled as: one-dimensional resisting structures. Because they can be described by a straight or curved line, along which the stresses channel the loads (Lines are said to have only one dimension, because a single number define the position of point on a line).

One-dimensional resisting elements may be used to cover a rectangular area, but such an arrangement is usually impractical and inefficient. For example, a series of beams, all parallel to one of the sides of the rectangle, serves this purpose, but a concentrated load on such a system is carried entirely by the beam under the load, and all other beams remain unstressed. This system is impractical—because one beam deflects, while the other beams remain horizontal, and inefficient, because it does not work as a whole in carrying the load. The load transfer occurs always in the direction of the beams, and the loads are supported by the two walls at their ends, while the walls parallel to the beams remain unloaded. This is a proper solution when unloaded walls are needed for functional purposes, but it becomes uneconomical, when the four walls enclosing the space can be used to support loads. (Figure 2.1.)

Figure 2.1: Concentrated Load on One-Dimensional System
The above consideration suggest that it would be structurally more efficient to have "two-way load dispersal." Such dispersal is obtained by means of grids and plates, two-dimensional resisting structures acting in a plane.

If two identical, simply supported beams, at right angles to each other, are placed one on top of the other, and a concentrated load is applied at their intersection, the load is transferred to the supports at the ends of both beams and dispersed in two directions. This mechanism involves the loading of the upper beam and its consequent deflection, the deflection of the lower beam by the action of the upper one, and the distribution of the load carrying action between the two beams — resulting in equal deflections because of compatibility between the two intersecting members. And since they are identical, each must carry half the load. Hence, each support reaction equals one-fourth of the load, and "two-way dispersal" reduces the loads on the supports to one-half the value, they would have, if only one-way dispersal took place. (Figure 2.2.)

![Square Plan Shape](image1)

![Load-Sharing Mechanism](image2)

**Figure 2.2.: Two-Way Load Dispersal by Equal Beams**
Two beams at right angles must deflect by the same amount at their intersection, even if they have different lengths or different cross sections. However, a greater load is required to deflect a stiffer beam as much as a more flexible one. Hence, the stiffer beam will absorb a greater share of the load than the more flexible beam, and the loads on the two beams will not be equal. The stiffness of a beam under a concentrated load is inversely proportional to cube of its length. Thus if two beams of identical cross section have spans in the ratio of one to two, their stiffnesses are in the ratio of eight to one. Consequently the short beam will carry eight-ninths of the load and the long beam one-ninth of the load. (Figure 2.3.)

![Figure 2.3: Two-Way Load Dispersal by Different Beams on Rectangular Plan Shape](image-url)
This study shows that the two-way transfer is efficient if the two beams are of equal, or almost equal, stiffness. As soon as one beam is much stiffer than the other, the stiffer beam carries most of the load, and the transfer occurs essentially in one direction. To obtain an efficient two-way transfer in the case of unequal spans, the longer beam must have a substantially stiffer cross section, that is, a greater moment of inertia.

The sharing mechanism or phenomenon of one concentrated load between two beams may be extended to a series of loads by setting a beam on two or more perpendicular beams, one under each load. Once again, the intersection points must deflect by the same amounts. In the Plan-arrangement of the deflections of the unsupported long beam at the quarter points would be smaller than the deflection at midspan. Hence, if the three shorter cross-beams have equal stiffness, the middle beam takes a greater share of the load than the side beams, since it deflects more. For beams of identical cross section, and cross-beams one-half as long as the long beam, the loads at the supports have the values. The greater dispersal in the direction of the shorter, stiffer span is evident, the long sides support 94 per cent of the total load. With a long beam eight times, stiffer, the long sides still carry about 65 per cent of the total load. (Figure 2.4.)

![3D View](image)

**Figure 2.4:** Two-Way Dispersal of Three Loads on Rectangular Plan Shape
Finally, one can cover a rectangular area by a grid of beams at right angles, and obtain the two-way dispersal of loads located at any grid intersection. If all the beams of the grid running in one direction are set above the beams running in the other, any one of the upper beams acts as a continuous beam on flexible supports provided by the lower beams. In this grid an uncentered load may produce upward deflection of an upper beam, and lift it from its supports, due to rigid connections introduced between the two members, obtained by means of welding etc.

Also rigid connections between the beams evolving grid-systems introduce another structural action in the grid. When two beams connected at midspan are deflected by a concentrated load at their intersection, their midspan sections (a) move downward, but remain vertical because of symmetry. When the intersection of the two beams does not occur at midspan, the beam sections (b) deflect and rotate. The continuity introduced by the rigid connection transforms the bending rotation of one section into a twisting rotation of the other. The stiffness of the grid is substantially greater when the beams are rigidly connected — than when they are simply supported, one over the other; this indicates that the twisting mechanism is capable of transferring part of the load, thus producing smaller displacements in the grid (Figure 2.5)
2.4.2 Finite Element Method: Introduction

In this section, emphasis will be on the study of literature pertaining to the Finite Element Method, which is based upon the general principle: from part to whole. This principle is not only applied in engineering analysis, but also in other spheres of human-life endeavour like humanities, social sciences, and physical sciences etc. And further inductive-reasoning based principle such as: particular to general, and from individual to universal is attempted for extended study patterns, logically.

In other words, the application of the principle is: attempting to obtain information about the whole, by understanding the parts, in the beginning, in general. Since in engineering-study, closed-form solutions are hardly possible within the means available. Therefore it becomes essential to consider physical medium: surfaces and bodies, as an assemblage of many small parts for framing background orientation, regarding the same. Discussing the same in the following, forms a first-step towards a meaningful study — which is attempted subsequently:
2.4.2.1 Finite Element Method: An Appraisal

(1) Overview

Finite element method is basically a numerical method used to analyse continuous systems. In brief, the basis of the method is on representation of the elastic continuous (structure) by an assemblage of sub divisions called finite elements. These elements are considered to be interconnected at joints, which are called nodes or nodal points. A displacement functional, such as poly or trigonometrical functions, represents the variation of displacement, which are unknowns. Hence the final solution yields the approximate displacements at the discrete points. Generally, polynomials are chosen to express the displacement functional, since they offer an ease in mathematical manipulation.

A variational principle of mechanics, such as principle of minimum potential energy or virtual work, is employed to obtain the set of equilibrium equations for each element. The potential energy of a loaded elastic body or structure is represented by the sum of the internal energy stored, as a result of the deformations and the potential energy of the external loads. If the body is in a state of equilibrium, this energy is minimum.

The equilibrium equations for the assembly (body) are obtained by combining the equations of the individual elements such that compatibility conditions are satisfied at the interconnected nodes. These equations are modified for the given displacement boundary conditions, and then solved to obtain the unknown displacements. Obviously the theory of finite elements consists of the study of the element in first phase, and that of the assemblage in the second phase.

In the process of discretising or subdividing an continuum the convergence of results is ensured. Discretisation pattern depends upon the geometry of the body or structure, and upon the number of independent space co-ordinates (e.g., x, y, or z) which are necessary to describe the problem. A finite element usually has a simple one, two or three dimensional configuration. The boundaries of the elements are oftenly straight lines. Although, for the problems defined in curvilinear co-ordinates elements, curved boundaries may be used, in specific cases.
2.4.2.1.1 Applications of Finite Element Method

The following steps are used in applications of finite element method, which are summarily reviewed here as under:

(1) **Discretisation of Continuum**

The given body is divided into an equivalent system of finite elements. The finite elements may be of any model e.g. triangular, trapezoidal, isoparametric, etc. The elements are assumed to be connected to each other at nodal points only.

(2) **Selection of Displacement Models**

A polynomial is more commonly selected as displacement functional to represent displacement at any point within the element in terms of the nodal displacements. Polynomial is preferred as displacement functional—as it permits differentiation and integration with ease, and its ease in adaptability to give desired approximation, by changing its order.

(3) **Derivation of Stiffness Matrix using a Variational Approach**

The stiffness matrix consists of the co-efficients of equilibrium equations derived from material and geometric properties of an element. The distributed forces applied to the structure are converted into nodal force vector \( \{Q\} \).

The nodal force vector \( \{Q\} \) and nodal displacement vector \( \{q\} \) are expressed as a set of simultaneous linear algebraic equations.

\[
[K][q] = \{Q\}; [K] = \text{Stiffness Matrix}
\]

(4) **Assembly of algebraic equations for overall discretized continuum**

The overall equilibrium relation between the total stiffness matrix \( [K] \), the total load vector \( \{R\} \) and the nodal displacement vector for the entire body \( \{r\} \) will expressed as a set of simultaneous equations.

\[
[K]\{r\} = \{R\}
\]

(5) **Solutions for unknown displacements**

The algebraic equations are solved by known mathematical techniques by using proper boundary conditions to determine the unknown displacements.

(6) **Computation of Element Strain & Stresses, from the nodal displacements**

Once the nodal displacements are known, the strains and stresses for each the element are determined as these values are related to the derivatives of displacement.
2.4.2.1.2 Advantages

Like all numerical approximations, the finite element method is based on the concept of discretisation. Nevertheless, as either a variation or residual approach, the technique recognises the multi-dimensional continuity of the body. Not only does the idealisation portray the body as continuous, but it also requires no separate interpolation-process to extend the approximate solution, to every point within the continuum.

Despite the fact—that solution is obtained at a finite number of discrete node points, the formulation of field variable model inherently provides a solution at all other locations in the body. In contrast to other variational and residual approaches, the finite element method does not require trial solution—which must apply to all the entire multi-dimensional continuum. The use of separate subregions, or finite elements, for the separate trial solutions—thus permits a greater flexibility in considering continua for complex shapes.

Most important advantage of finite element method is derived from the techniques of introducing boundary condition. This is another area in which the method differs from other variational or residual approaches. Rather then requiring every trial solution to satisfy the boundary condition, one prescribes the conditions after obtaining the algebraic equations for the assemblage. Since the boundary conditions do not enter into equations for the individual finite element, one can use same field variable models for both internal and boundary elements. Moreover, the field variable model need not be changed, when the boundary conditions change.

The introduction of boundary condition into the assembled equation is a relatively easy process. Here only the geometric boundary conditions need be specified in a variational approach, because the natural conditions are implicitly satisfied. No special techniques or artificial devices are necessary, such as the non-centred difference equations or fictitious external points, often employed in the finite difference method.

The finite element method not only accommodates complex geometry and boundary conditions, but it also has proven successful in representing various types of complicated material properties—that are difficult to be incorporated into other numerical methods. For example, formulations in solid mechanics have been devised for anistropic, Nonlinear, hysteretic, time-dependent, or temperature-dependent material behaviour.

Also, finite element method accounts for non-homogeneity by the simply tactic of assigning different properties to different properties to different elements, which used to be difficult in other numerical formulations.
Further, Finite Element Method has proved to be a powerful and versatile tool, for a wide range of problems by following systematic steps involved in its applications.

Among many packages, selected references are being mentioned here like: ASKA, STRUDL, SAP, NASTRAN, STAAD-III, ANSYS (5.3) and SAFE etc. Another indication of the generality of the method is that programs developed for one field of engineering, have been applied successfully, to problems in other fields with little, or no modification.

Finally, an engineer may develop concept of finite element method at different levels. It is possible to interpret the method in physical terms. On the other hand, the method may be explained entirely in mathematical term. The physical or intuitive nature of the procedure is particularly useful to the engineering student and practicing engineer. Nevertheless, it is significant that the method has mathematical foundations.
2.4.2.1.3 Limitations

One limitation of the finite element method is that a few complex phenomena are not accommodated adequately by the method at its current state of development for example: cracking and fracture behaviour contact problems, bond failure of composite materials, Also seepage problems are another area not adequately covered presently. No doubt the finite element method has reached a high level of development as a solution technique but the method yields realistic results only, if the coefficients or material parameters which describe the basic phenomena are available — Material nonlinearity in solid mechanics is another area, also not covered adequately.

In order to exploit fully the power of finite element method, significant efforts must be made to develop constitutive laws, and the evaluation of realistic coefficients and material parameters. Even the post efficient computer code require a relatively large amount of computer memory and time use restricted to those having, large and high-speed computers.

Another important and fundamental limitation is in generating error — free input data for the computer, because of time consumed in basic process of subdividing the continuum for the computer. Although this process may be made automatic, yet engineering judgement is employed in the discretisation, so essentially needed. Even sound judgement is needed in interpretation of computer results. Consequently, it is essential that engineer or programmer provide checks, to detect such errors for effective results.

Finally, like any other approximate numerical methods, the results of a finite element analysis must be interpreted with care. We must be aware of assumptions employed in the formulation, the possibility of numerical difficulties, and the limitations in the material characterizations used. A large volume of solution information is generated by a finite element routine, but this data is worthwhile only, when its generation and interpretation are used by proper-engineering-judgement.

2.4.2.1.4 Conclusions

Since 1971, the finite element method has attained an status of maturity as analysis technique. Future developments will consist of refinement of the mesh-techniques, and a broader range of applications — rather than major changes in the analysis procedure.

2.4.2.2 Convergence Requirements

In any acceptable numerical formulation the numerical solution must converge to exact solution of the problem. For the finite element method, it has been found that under certain circumstances the displacement formulation provides an
upper bound to the true stiffness of the structure or in other words, the stiffness coefficients for a given displacement model have magnitudes higher than those for the exact solution. If, the finite element subdivision of the structure is made thinner, the approximate displacement solution will bound the exact solution from below. In order for this boundedness to be assumed, three conditions given below should be satisfied:

1. The displacement models must be continuous with in the elements and the displacements must be compatible between adjacent elements.

   The first part of this requirement is readily met by choosing polynomial models, which are inherently continuous. The second part implies that adjacent elements must deform without causing openings overlaps or discontinuities between the elements.

2. The displacement models must include the rigid body displacement of element.

   A rigid body displacement is most elementary deformation that an element may undergo. Basically this condition states that there should exist combinations of values of generalized coordinates that cause all points on element to experience the same possible rigid body translations and rotations.

3. The displacement models must include the constant strain states of the elements.

   This requirement can be stated in terms similar to those of the second conditions. There should exist combinations of values of generalized co-ordinates that cause all points on element to experience the same strains. One such combination should occur for each possible strain. The necessity of this requirement can be understood physically if we subdivide a body into smaller and smaller elements. As these elements approach infinitesimal size, the strains in each element approach constant values unless our approximation include these strains, we cannot hope to converge to a correct solution.
2.4.2.3 STAAD-III for Windows / Windows NT: Literature Reviewed

(1) Introduction

STAAD-III for Windows/Windows NT is a comprehensive structural engineering software — that addresses all aspects of structural engineering-model development, analysis, design, verification and visualization. STAAD-III for Windows/Windows NT is based on the principles of "concurrent engineering". One can build the model, verify it graphically, perform analysis and design, review the result, sort and search the data to create a report — all within the same graphics based environment. With a live relational database at its core, the Windows based Graphical User Interface controls, and manages all the functions.

Following are the main options available from the Concurrent Graphics Environment:

(2) STAAD-PRE Graphical Input Generation

STAAD-III performs the analysis and design of the structure. The processes of analysis and design are integrated and can be performed in the same run. STAAD-III uses a command language based input format which can be created through an editor, the powerful STAAD-PRE graphic input generator or through CAD based input generators. Output generated by STAAD-III consists of detailed numerical results for analysis/design, and sharp presentation quality printer plots of the run document.

(3) STAAD-POST Graphical Post Processing

STAAD-POST is a powerful graphics facility for verification of the model and display of the results. The model verification capabilities include complete graphics verification and visualisation for all items. State of the art results verification capabilities include display and plotting of structure geometry, deflected/mode shapes, bending moment/shear force diagrams, stress-contours etc. A versatile "query" facility allows generation of customized reports. Powerful icon based graphics tools provide extremely user friendly navigation and manipulation capabilities.

The following list shows the suggested project development sequence for solving problem with STAAD-III. The order is quite flexible.

(a) Create the STAAD-III input file with the Edit Input option, or with the STAAD-PRE graphical modelling option. Also one can download/import files from CAD software, by using the STAAD-PRE option.

(b) Enter the STAAD-PRE option with partially complete input files or the STAAD-POST option with complete input files to check the model graphically.
(c) Perform the analysis and design with the STAAD-III option on the main menu screen or with Run STAAD icon the toolbox in the STAAD-POST environment.

(d) Verify the analysis results and designs reported the the text output file. This can be done with the VIEW OUTPUT option on the main screen or with the RESULT and QUERY items in the STAAD-POST option.

(e) Verify the analysis and design results graphically with the Result, Report, Query and Icon items in the STAAD-POST option.

(f) Generate necessary plots using the STAAD-POST facility.

(4) Loads

Various loads on a structure can be applied in the form: Joint Load, Temperature Load, and Fixed-End Member Load. STAAD-III can also generate the self-weight of the structure and use it as uniformly distributed member loads in analysis. Any fraction of this self-weight can also be applied in any desired direction.

(a) Member Load

Three types of member loads may be applied directly to a member of a structure. These loads are uniformly distributed loads, concentrated loads, and linearly varying loads (including trapezoidal). Uniform loads act on the full or partial length of a member. Concentrated loads may act over the full length of a member at a specific location(s). Trapezoidal linearly varying loads may act over full or partial length of a member.

(b) Fixed-End Member Load

Load effects on a member may also be specified in terms of its fixed end loads. These loads are given in terms of the member coordinate system and the directions are opposite to the actual load on the member. Each end of a member can have six forces: axial y; shear z; torsion; moment y; and moment z.

(c) Temperature/Strain Load

Temperature difference through the length of a member as well as differences of both faces of members and elements many also be specified. The program calculates the axial strain (elongation and shrinkage) due to the temperature difference. From this it calculate the induced forces in the member and the analysis is done accordingly. The strain intervals of elongation and shrinkage can be input directly.

(d) Support Displacement Load

Loads can be applied to the structure in terms of the displacement of the supports. Displacement can be translational or rational. Translational displacements
are provided in the specified length while rotational displacements are always in degrees. Note that displacements can be specified only in directions in which the support is restrained and not in directions in which it is released.
2.4.2.3.1 STAAD-III: General

(1) Introduction

This section contains a general description of the analysis and design facilities available in STAAD-III. The objective of this section is to familiarise the user with the basic principles involved in the implementation of the various analysis/design facilities. As a general rule, the sequence in which the facilities are discussed follows the recommended sequence of their usage in the input file.

(2) Input generation

The user communicates with STAAD-III through an input file. The input file is a text file consisting of a series of commands which are executed sequentially. The commands contain either instructions or data pertaining to analysis and/or design.

The STAAD-III input file can be created through a text editor or the STAAD-PRE input generation facility. In general, any text editor may utilise to create the input file. The input generation facility creates the input file through an interactive menu-driven graphics oriented procedure.

(3) Programme Capabilities

Almost any type of structure can be analysed by STAAD-III. Most general is the SPACE structure, which is a three dimensional framed structure with loads applied in any plane.

(4) Structure Geometry and Coordinate Systems

Structure is a assembly of individual components such as beams, columns, slabs plates etc.. In STAAD-III frame elements and plate elements may be used to model the structural components. Typically, modelling of the structure geometry consist of two steps:

(a) Identification and description of joints or nodes.
(b) Modeling of members or elements through specification of connectivity (incidences) between joints.
(c) In general, the term Member will be used to refer to frame elements; and the term Element will be used to refer to plate/shell elements. Connectivity for Members may be provided through the Member Incidence command while connectivity for Elements may be provided through the Element Incidence command. STAAD-III uses two types of coordinate systems to define the structure geometry and loading patterns. The Global Coordinate System is an arbitrary coordinate system in space which is utilised to specify the overall geometry and loading pattern on the structure. A Local Coordinate System is associated with each
member or element and is utilised as Member End Forces output on local load specification.

(5) Finite Element Information

STAAD-III is equipped with a state of the plate-shell and solid finite element. The features of each is explained as.

(6) Plate/Shell Element

The plate bending portion can handle thick and thin plate, thus extending the usefulness of the plate elements into a multiplicity of problems. In addition, the thickness of the plate is taken into consideration in calculating the out of plane shear.

The plane stress triangle behaves almost on par with the well known linear stress triangles of most similar flat shell elements incorporate the constant stress triangle which has very slow rates of curvature where the quadrilateral element may not be suitable.

(7) Element Numbering

The Plate/Shell finite element is based on the hybrid element formulation. The elements may be quadrilateral also. If all the four nodes of a quadrilateral element do not lie on one plane, it is advisable to model them as triangulate elements. The thickness of the elements may be different, from one node to another.

Surface structure such as wall, slabs, plates and shells may be modelled using finite elements. For convenience in generation of a finer mesh of plate/shell elements within a large area, Mesh Generation facility is also available.

The user may also use the element for Plane Stress action only. The Element, Plane Stress command should be used for this purpose.

(8) Geometry Modelling Considerations

(a) The program automatically generates a fifth node "O" at the element centre.

(b) While assigning nodes to an element in the input date, it is essential that the nodes be specified either clockwise or counter clockwise. For better efficiency, similar elements should be numbered sequentially.

(c) Element aspect ration should not be excessive. They should be on the order of 1:1 and preferably less than 4:1.

(d) Individual elements should not be distorted. Angles between two adjacent element sides should not be much larger than 90° and never larger than 180°.
(9) Element Load Specification

Following load specifications are available

(a) Joint loads at element nodes in global directions.

(b) Concentrated loads at any user specified point within the element in global or local directions.

(c) Linearly varying pressure on element surface in global or local directions.

(d) Partial uniform pressure on user specified portion of element surface in global or local directions.

(e) Linearly varying pressure on element surface in local directions.

(f) Temperature load due to uniform increase or decrease of temperature.

(g) Temperature load due to difference in temperature between top and bottom surfaces of the element.
2.4.2.4 The ANSYS Program: Literature Reviewed

The ANSYS program is designed to be user oriented. It does not require special knowledge of system operations or computer programming in order to be used. The basic input is straightforward and easily learned. The major complexities arise from the fact that the program has numerous capabilities and options. In addition, the program flexibility allows various methods of arriving at the same solution. The flexibility, capabilities, and options have been developed over many years, at the request of a worldwide user community, such that the ANSYS program can be applied to a wide variety of engineering applications.

No one user is expected to know and fully understand the entire ANSYS program before attempting a solution. In fact, the opposite is usually true i.e. the user often learns about an option because of a particular need in the solution. The program is structured—such that the special purpose options are “off” and the data input defaults are set to basic logical values—so that the user should not be concerned about the more complex paths of ANSYS—unless these paths are purposely selected.

The ANSYS program is a self-contained general purpose finite element program developed and maintained by Swanson Analysis Systems, Inc. The program contains many routines, all inter-related, and all for the main purpose of achieving a solution to an engineering problem by the finite element method. Once the ANSYS program is accessed, the user may logically progress through the desired capabilities. Even some common operations, such as file rewinds, file copies and file listing can be accomplished within the ANSYS program.


An Engineering problem is usually solved in three phases: (1) Preprocessing (2) Solution and (3) Postprocessing. Refer Table 2.1 (shown below) for selected operations, in each phase. The postprocessing phase is optional, but is highly recommended for reducing, reorganising, and interpreting the solution output.

The main routines and analysis types associated with these phases are given in Table 2.2 (Given in the end). Routine and analysis types are not mutually exclusive. For example, preprocessing may consist of using the PREP7 routine alone, or for some transient analyses both PREP7 and PREP6 may be used. In general PREP 7 is used for the majority of all preprocessing.
Once the Pre-processing phase has been completed, the user may progress through various analyses, (on the same model) in the solution phase.

Once the solution phase has been completed the user may progress through various postprocessing operations (on the same solution output). For example, POST1 may be used to process the basic solution data, followed by POST27 for various solution combinations, followed again by POST1 for processing the combined solution data.

### 2.4.2.4.1 Computer Processing Systems

The ANSYS program is available on most commercial computers used for engineering analysis ranging from personal computers (PC) through large mainframe computers. Special ANSYS PC versions (with limited preprocessor, solution, and postprocessing capability) are available for various PC products. On mini computers and mainframes, the entire program may be installed (giving full preprocessing, solution, and postprocessing capabilities) or, in some cases, reduced versions may be installed.

All versions accept the same basic input data commands and certain output files may be input to other versions for distributed processing. Preprocessing output files are coded files, and may be transmitted directly from machine. Solution output files are machine dependent binary and must be coded before transmitted.

<table>
<thead>
<tr>
<th>Preprocessing Phase</th>
<th>Solution Phase</th>
<th>Postprocessing Phase</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mesh Generation</td>
<td>Element Matrix</td>
<td>Post Solution Operations</td>
</tr>
<tr>
<td>Geometry Definitions</td>
<td>Formulation</td>
<td>Post Data Printout (For Reports)</td>
</tr>
<tr>
<td>Material Definitions</td>
<td>Overall Matrix</td>
<td>Post Data Scanning</td>
</tr>
<tr>
<td>Constraint Definitions</td>
<td>Triangularisation (Wave Front)</td>
<td></td>
</tr>
<tr>
<td>Load Definitions</td>
<td>Displacement, Stress, Etc., Calculations</td>
<td></td>
</tr>
<tr>
<td>Model Display</td>
<td></td>
<td>Post Data Display</td>
</tr>
</tbody>
</table>

**Table 2.1: Typical Phases of analysis**
2.4.2.4.2 Interactive Vs. Batch Runs

The ANSYS program is designed to run fully batch, fully interactive, or a combination of both. The data input is essentially the same for either mode of running. If interactive communication with the computer is available (along with on-line graphics display capability) it is recommended that the preprocessing phase be done interactively, the solution phase be submitted as a batch job, and than the postprocessing phase be done either interactively or batch.

(1) Interactive Runs

Interactive running has advantages over batch running in that it is self-teaching, allows less dependence on the User's Manual, produces immediate displays and output, and allows immediate model editing and corrections. The disadvantage is that "thinking time" may be charged at a premium rate while connected to the host machine.

Interactive input is like batch input, with the following exceptions: (a) The first ANSYS command of an interactive run must be /INT (not necessary when the program execution command activates the /INT command); (b) A user-interface menu system with "mouse" picking may be available (terminal dependent) to help the user in data preparation. A general help facility (HELP) is also available independent of the user-interface.

An interactive run may proceed as follows:
(a) Execute the PREP version, the interactive version, or the full ANSYS program according to system procedure.
(b) Select interactive mode. After a system response indicating the start of execution (if any), input /INT. Any other command input at this point will put the program into the batch mode. If this occurs, enter the /INT command.
(c) Input each data command as desired. Any command entered at this point that is not recognizable will be ignored.
(d) Interactive guidance is available. A user-interface menu system may be used to prepare data and to obtain user assistance and on-line documentation (terminal dependent). The menu system is activated by the documentation (terminal dependent). The menu system is activated by the /MENU, ON command after graphic device is defined (/SHOW) and is deactivated by the /MENU, OFF command. Activating the menu system creates a graphic display area. "Mouse" picking may be used for data entry. Enter HELP, MENU for details. The System is self-documenting with numerous help screens. A "Introduction to the User Interface" document explaining the user interface and its particular commands is also available.
The HELP command is also available for descriptions of ANSYS commands in general. The STATUS command is available within each module for a summary of the current specification settings.

(e) Immediate graphics displays are also available during an interactive session. After the first display is formed, the terminal screen (terminal dependent) is divided into a graphics screen and a dialog screen. The /G, /T, and /B commands may be used to toggle among the graphics screen only, standard text screen, or both (graphics and dialog screens) respectively.

(f) If an error causes an abnormal exit from ANSYS, or if the user does a deliberate program abort, control will revert back to the system level. Re-enter ANSYS as described in Step 1. A file of all previous ANSYS input commands is written as File 18 during the run up to the "abort" point. This file may be edited and re-input to return to the point before the exit.

(g) Normal program exit. A/EOF command input at any Routine Begin level will cause a normal exit from the ANSYS program.

(2) Batch Runs

If the interactive mode is not available, or desired, the user may proceed as follows:

(a) Read over the applicable sections of the ANSYS User's Manual and review the sample problems.

(b) Prepare data file with a text editor. The data may consist of a single phase or all phases of the analysis (back-to-back). A partially generated model may be all that is to be prepared at this time.

(c) Submit the file for the batch run.

(d) Recover printout and graphics displays produced from the run.

(3) Related Terminology

Command, Field, Label, Data, Line, Display or Plot, Module, Routine, Program, Run, Analysis, Preprocessing Phase, Solution Phase, Post Processing Phase.

(4) Analysis Types
Table 2.2: Basic ANSYS Routines or Analysis Types Associated with the Three Phases

(5) Typical Analysis Template

(a) ENTER THE ANSYS PROGRAM (System Command)

(b) BUILD THE MODEL

- Specify job name (/FILNAME)*
- Specify analysis title (/TITLE)*
- Record system of units (/UNITS)*
- Enter PREP7 (/PREP7)
- Define element types and options (ET or control panel)
- Define element real constants (R or control panel)**
- Define material properties (MP)
- Create model geometry (solid modelling or direct generation)
  - Build model
  - define meshing controls (ESIZE, ESHAPE, etc.)
  - save the database (SAVE)*
  - mesh (Xmesh)
  - Exit PREP7 (FINISH)
(c) APPLY LOADS AND OBTAIN THE SOLUTION

- Enter SOLUTION (/SOLU)
- Define analysis type and options (ANTYPE or control panel)**
- Apply loads
  - DOF constraints (D.DK)
  - Force loads (F,DF)
  - Surface loads (SF, SFL)
  - Body loads (BF)
  - Inertia loads (ACEL, OMEGA, IRLF, etc.)
- Save the database (SAVE)*
- Initiate the solution (SOLVE)
- Exit SOLUTION (FINISH)

(d) REVIEW THE RESULTS

- Enter POST1 (/POST1)
- Resume the database (RESUME)**
- Read results date from results file (SET)**
- Display results (PLDISP, PLNSOL, PLESOL, etc.)
- List results (PRNSOL, PRESOL, PRETAB)
- Exit POST1 (FINISH)

(e) EXIT THE ANSYS PROGRAM (/EXIT)

---

Note: * This step is optional but recommended
** This step may not be required
Advancing technology provided the builder with new materials and more efficient methods which were often in glaring contrast to our traditional conception of architecture. I believed, nevertheless, that it would be possible to evolve an architecture with these means. I felt that it must be possible to harmonize the old and the new in our civilization. Each of my buildings was a statement of this idea and a further step in my search for clarity.

— Ludwig Mies van der Rohe
(A Personal Statement, 1964)