CHAPTER 1

INTRODUCTION

1.1. WHAT IS BUCKLING?

In general, there are two major categories leading to the failure of a mechanical component: material failure and structural instability, which is often called buckling. The term buckling is associated with a (observable) process whereby a given state of a deformable structure suddenly changes its shape. Brought about by a varying external load, this change in configuration often happens in a catastrophic way. The change of shape observed during the buckling process is usually very pronounced and it occurs without a warning. Another important characteristic is that even in carefully monitored experiments, the beginning of failure (if it occurs) cannot be predicted with precision. A good set of examples of buckling phenomena encountered in the engineering practice can be found in Bushnell (1985).

For a common man, the word ‘buckling’ means sudden catastrophic failure of a structure involving large deformations. But in engineering parlance, the buckling is a phenomenon that generally occurs well before deformations are very large. When a slender structure is loaded in compression, for small loads it deforms with hardly any noticeable change in geometry and load-carrying capacity. On reaching a critical load value, the structure suddenly experiences a large deformation and it may lose its ability to carry the load further. At this stage, the structure is considered to have buckled. For example, when a rod is subjected to an axial compressive force, it first shortens slightly but at a critical load the rod bows out and we say that the rod has buckled. In the case of a circular ring under radial pressure, the ring decreases in size slightly before buckling into a number of circumferential waves. For a cruciform column, under axial compression, it shortens and then buckles in torsion.

For small loads the process is elastic since buckling displacements disappear when the load is removed. Local buckling of plates or shells is indicated by the
growth of bulges, waves or ripples, and is commonly encountered in the component plates of thin structural members.

Buckling and bending are similar in one way as both involve bending moments. In bending, these moments are substantially independent of the resulting deflections, whereas in buckling, the moments and deflections are mutually interdependent - so moments, deflections and stresses are not proportional to loads (Figure 1.1). If buckling deflections become too large then the structure fails - this is a geometric consideration, completely divorced from any material strength consideration.

![Comparison between bending and buckling processes](image)

**Figure 1.1.** Comparison between bending and buckling processes

Of late, buckling has become a major structural problem because of the use of lesser amount of high strength material for load support in structural applications towards weight consideration. As a result, more structures and component plates / shells thereof have become generally slender and hence prone for buckling requiring their design to satisfy both strength as well as buckling safety constraints.

### 1.2. WHAT IS THIN SHELL?

In structural analysis, a thin shell may be defined as a body in which the distance from any point inside the body to the reference middle surface is small in comparison with any typical dimensions of the reference surface such as a radius of curvature [Brush and Almroth (1975)] i.e., the thickness of shell always being considered small in comparison with other dimensions of the shell and with its radii of
curvature. A shell has two radii of curvature at every point and each is measured normal to the shell mid surface and each is radius of a small arc drawn on the shell mid surface. Familiar examples of shells are an eggshell and a water tank.

Typically, the thickness dimension of the plate is much smaller than its planner dimensions yielding a “thin-walled” type of structure. A plate is said to be thin plate when the ratio between the length \((a)\) and thickness \((t)\) of the plate [namely the \(a/t\) ratio] is approximately in the range of \(20 – 100\), in which only membrane and bending deformation effects are to be considered in the analysis. When \(a/t\) ratio exceeds 100, it is a very thin plate in which the membrane effect will be dominant and it would be treated as a geometrically nonlinear shell. Similarly, when the \(a/t\) ratio is less than \(20\), the plates are designated as thick plates in which the effect of shear stress distribution in the transverse direction should also be included in the analysis, in addition to the membrane and bending deformation effect [Wang et al. (2009)].

1.3. WHY DO THIN SHELLS BUCKLE?

These thin shells have two types of stiffness - membrane stiffness and bending stiffness. The membrane stiffness is, in general, several times greater in magnitude than the bending stiffness. A thin shell can absorb a large amount of membrane strain energy without deforming too much. To absorb equivalent amount of bending strain energy, the shell has to deform much more than to store in the form of membrane strain energy.

If a shell is loaded in such a way that most of its strain energy is in the form of membrane compressive energy and on reaching a critical load condition, the shell finds a way to convert the stored up membrane energy into bending energy. This process of exchange of stored up membrane strain energy into bending energy is called “Buckling”. Very large deflections are generally required to convert a given amount of membrane energy into bending energy. The way in which buckling occurs depends on how the shell is loaded and also on the geometry of the shell structure and its material properties.
1.4. TYPES OF BUCKLING BEHAVIOUR

Koiter is the first one, who formulated the hypothesis that all real world structures have some form of small initial imperfections, in spite of how carefully they were manufactured, and these small unavoidable imperfections cause the large differences between theoretical and experimental results. From Koiter's asymptotic theory, three different types of nonlinear post-buckling behaviours of thin walled structures can be recognized as shown in Figure 1.2 (unstable component of buckling behaviors are shown in dotted lines), namely, stable symmetric, unstable symmetric, and asymmetric. In the pioneering work of Koiter (Bushnell, 1985), distributed axisymmetric imperfections models were used to determine the shell sensitivity to initial geometrical imperfections.

The stable symmetric post buckling behaviour shown in Figure 1.2 (a) is typical of an axially compressed isotropic plate. If the perfect plate is loaded precisely in its neutral surface it buckles either way with equal ease and post buckling equilibrium state is stable. The unstable symmetric post buckling behaviour shown in Figure 1.2 (b) is typical of axially compressed thin cylindrical shells and externally pressurized thin spherical shells. The asymmetric post buckling behaviour shown in Figure 1.2 (c) indicates that the structures continue to carry the loads above the bifurcation load of perfect structures if they are forced to buckle in one way, but collapse if they are allowed to buckle in the other way. One of the examples for such structural behaviour is a built-up panel consisting of a flat sheet riveted to a corrugated sheet.

![Different types of post-buckling behavior](image)

**Figure 1.2.** Different types of post-buckling behavior [Bushnell (1985)]
The unstable symmetric and asymmetric cases are often referred to as imperfection sensitive since small imperfections can cause a drastic decrease in the predicted critical load. Researchers have also discovered that the critical points for the unstable symmetric and asymmetric cases degenerate to limit points when imperfections are introduced.

Since plate structures show stable symmetric post buckling behavior, its ultimate load carrying capacity is decided by the collapse load of the structure which in turn depends on the initial imperfections present in those structures. (Figure 1.3.)

1.5. STIFFENED STEEL PLATES UNDER UNIAXIAL COMPRESSION

Most plated structures, although quite capable of carrying tensile loadings, are poor in resisting compressive forces. Usually, the buckling phenomena observed in compressed plates take place rather suddenly and may lead to catastrophic structural failure. Therefore it is important to know the buckling / ultimate capacities of the plates in order to avoid premature failure.

Thin steel plates that are stabilized in one direction by stiffeners are used extensively for plating of deck and bottom of ship structures, plate and box girders of bridges, components of offshore structures, bridge decks, land-based structures such
as bins and bunkers and structures used in aerospace industries in which a high strength-to-weight ratio is important. The analysis of typical stiffened plate structure can be performed at grillage level, stiffened panel level between two adjacent transverse stiffeners, and bare plate element level between longitudinal and transverse stiffeners (Figure 1.4).

![Typical stiffened plate structure in a ship](image)

**Figure 1.4.** Typical stiffened plate structure in a ship

Local buckling and collapse of plating between stiffeners is considered as one of the basic failure modes. The bending resistance of the bounding edges of the bare plate in between longitudinal and transverse stiffeners is high compared with that of the plate itself. The rotational restraints along the plate edges can be considered small for plates subjected to axial compression; hence, the bare plate can be considered as simply supported along all edges [Suneel Kumar *et al.* (2007)]. Under normal operating conditions, the plates between stiffeners experience significant compressive loading during hogging and sagging movements of ships [Suneel Kumar *et al.* (2007)] and therefore determination of ultimate collapse strength of the steel plates is essential for safe design. Because of the presence of compressive axial forces and bending moments, stiffened plates are susceptible to failure by instability. Stiffened plates under uniaxial compression or under combined bending and compression can buckle in one of four forms: plate-induced overall buckling (PI), stiffener-induced overall buckling (SI), plate buckling (PB) and stiffener tripping (ST).

Overall buckling (sometimes referred to as Euler buckling or grillage buckling) is characterised by simultaneous buckling of the stiffener and the plate. If buckling occurs with the stiffener on the concave (compression) side of the plate,
overall buckling is said to be stiffener induced. This mode of failure is illustrated in Figure 1.5 (a), which shows a one-plate panel taken from a multiple panel assembly, consisting of one stiffener and the plate tributary to the stiffener. On the other hand, if the stiffener is on the convex side of the plate, overall buckling is said to be plate induced (Figure 1.5 (b)). These two types of failure mode are typically characterized by a very stable post-buckling response as shown in Figure 1.6 where buckling is characterized by a sharp change in stiffness.

Figure 1.5. Typical buckling modes [Sheikh et al. (2002)]

Figure 1.6. Load versus deformation behaviour [Sheikh et al. (2002)]
Plate buckling failure is characterized by buckling of the plate between the stiffeners, resulting in a load re-distribution from the plate into the stiffeners. This mode of failure is illustrated in Figure 1.5 (c) with a typical load versus displacement behaviour presented in Figure 1.6. The plate buckling failure mode tends to experience a more significant drop in load-carrying capacity in the post-buckling range than the overall buckling failure modes. Stiffener tripping is characterized by rotation of the stiffener about the stiffener-to-plate junction [Figure 1.5 (d)]. Stiffener tripping is, therefore, a form of lateral torsional buckling where torsion takes place about the stiffener-to-plate junction.

Of all of the failure modes, sometimes the plate buckling failure (i.e.,) failure of bare plate between stiffeners is a basic failure mode and also can be the most dangerous due to having a maximum strength lower than the other failure modes. Hence an unstiffened plate buckling is taken as the subject of study throughout this thesis.

1.6. BUCKLING OF UNSTIFFENED PLATE

Buckling behaviour of unstiffened plate is influenced by factors such as material and geometric properties, boundary and loading conditions, initial distortions, residual stresses and the degree of use. [Guedes Soares (1988a, & 1988b)]. The behaviour of plate panels under predominantly compressive loads usually undergoes five stages: pre-buckling, buckling, post-buckling, collapse (ultimate strength) and post-collapse.

In the pre-buckling stage, the plate’s response to load follows the Hooke’s Law where its load displacement relationship is linear. When compressive loads reach the “critical buckling load” of the plate, buckling occurs. Buckling strength can be defined as the end stress when the buckling profile of the plate (usually in the form of half-waves of approximately equal length) is first observed during incremental loading. Thin plates usually show elastic buckling, while thick plates usually exhibit inelastic buckling. A plate having buckled in the elastic region will eventually collapse, resulting in a rapid decrease in in-plane stiffness. On the other hand, if
buckling occurs in the inelastic region, plates normally reach the ultimate limit state immediately. In other words, the buckling and ultimate strength of the plate are equal in the inelastic range.

Theoretically, the buckling phenomenon of a plate structure may be described from a plot of the out-of-plane displacement \( (w) \) at a specific point, i.e. point of maximum displacement, against in-plane load \( (N) \), as shown in Figure 1.7.

![Figure 1.7. Out of plane displacement of a plate](image)

![Figure 1.8. Load-end displacement path](image)

When a perfect flat plate is subjected to low in-plane compressive loads, it remains flat and is in equilibrium condition. As the magnitude of the in-plane compressive load increases, however, the equilibrium configuration of the plate is eventually changed to a non-flat configuration and the plate becomes unstable. The magnitude of the compressive load at which the plate becomes unstable is called the “critical buckling load.” In classical linear buckling theory, when in-plane load increases from zero, an out-of-plane displacement remains zero, and a load-displacement curve follows Path 1 (Figure 1.7) until buckling load is reached. At this point, which is called a bifurcation point, the load-displacement curve may follow Path 2 which is a theoretical linear buckling path. Buckling load \( (N_{cr}) \) can be obtained from classical linear buckling theory. The critical buckling load is defined on this horizontal line. In a nonlinear theory, the curve follows Path 3, which is called a “post-buckling” curve. This curve is important in the study of plate behaviour beyond the buckling load. However, for a real plate with initial imperfections, the curve will
not follow Path 1, i.e. an out-of-plane displacement occurs as soon as load $N_x$ is applied. In this case, the load-displacement curve follows Path 4 from the beginning of the loading. Figure 1.8 shows the buckling behaviour of compressed plates in terms of edge (end) displacement.

### 1.7. INTRODUCTION TO FINITE ELEMENT ANALYSIS

To predict the buckling strength of the shell structures (either bifurcation buckling load or ultimate collapse load) analytical approaches or numerical approaches are generally followed. In the numerical approach, the Finite Difference Method (FDM) and Finite Element Method (FEM) are the mostly used ones. Among the numerical procedures, the finite element methods are the most frequently used in recent years.

Finite element analysis (FEA) is a method for numerical solution of field problems which can be either described by differential equations or by an integral expression. A field problem requires determination of the spatial distribution of one or more dependent variables. Finite element (FE) formulations, in ready-to-use forms, are available in general purpose FEA programs and hence, it is possible to use FEA programs, having little knowledge of the analysis. Although modern FE programs routines include significant library of inherent intelligence, diagnostics, and self correction, the analyst’s experience is still critical to the success.

Individual finite elements can be visualised as small pieces of a structure. In each finite element a field quantity is allowed to have only a simple spatial variation, e.g. described by polynomial terms up to $x^2$, $xy$ and $y^2$. The actual variation in the region spanned by an element is almost certainly more complicated, hence a finite element analysis provides an approximate solution. In many engineering situations today, we find that it is necessary to obtain approximate numerical solutions to problems, rather than exact closed-form solutions.
1.7.1. Brief history

The finite element method used today was developed to its present state very recently. The questions about the history of development of FEM, who originated the finite element method and when did it begin, have two different answers depending on asking to a mathematician or an engineer [Zienkiewicz (2002) and Stasa (1985)]. All of them have some justification to claim that the finite element method as their own, because each developed to get essential ideas of FEA independently at different times and for different reasons. However, the present day FEM does not have its roots in any discipline. Mathematicians are trying to improve the mathematical background of FEM, while the engineers are interested in applications where FEM can be used. In most branches of engineering, these developments have made the FEM as one of the most powerful numerical solution methods.

According to Zienkiewicz (2002), the development has occurred along two major paths, one in mathematics and other in engineering; somewhere in between these two paths are variational and weighted residual methods. Both of these require trial functions to effect a solution. The use of these trial functions is almost 200 years old. These trial functions are assumed, based on physical intuition and they are applied globally to get the solution for the problem. The use of trial functions is neither considered as development in pure mathematical field nor in engineering field. In 1795, Gauss used trial functions in what is now called as the method of weighted residuals. Later, Rayleigh used these trial functions in variational method in 1870 and by Ritz in 1909. In 1915, Galerkin introduced a particular type of weighted residual method which is called by his name as “Galerkin weighted residual method”.

The finite element method as we know it today seems to have originated with Courant in 1943 [Cook et al. (2002)], though he didn’t use the terminology “Finite element”. Courant determined the torsional rigidity of a hollow shaft by dividing the cross-section into triangles and interpolating a stress function $\phi$ linearly over each triangle from the values of $\phi$ at nodes. He introduced piecewise trial functions which are now called as shape functions. These shape functions are applied in a smaller region (i.e. at element level) instead of applying it globally which made him to solve the real world problems. In the early 1940s, aircraft engineers were developing and
using analysis method called force matrix method which is recognized as early form of finite element method. In this method, the nodal unknowns are forces not the displacements. When the displacements of each node are taken as unknown, the method is called as “Stiffness method”. The name finite element was coined by Clough in 1960. Many new elements for stress analysis were soon developed. In 1963, finite element analysis acquired respectability in academia when it was recognised as a form of the Rayleigh-Ritz method. Thus finite element analysis was seen not just as a special trick for stress analysis but as a widely applicable method having a sound mathematical basis. In 1965, Zienkiewicz and Cheung applied FEM, to solve non-structural problems. In 1969, Szabo and Leo showed how the weighted residual method, particularly the Galerkin method could be used in non-structural problem analysis.

The first textbook about finite element analysis appeared in 1967 [Zienkiewicz and Cheung (1967)] and today there exists an enormous quantity of literature about finite element analysis. General-purpose computer programs for finite element analysis emerged in the late 1960’s and early 1970’s. Since the late 1970’s, computer graphics of increasing power have been attached to finite element software, making finite element analysis attractive enough to be used in actual design. Previously it was so tedious that is was used mainly to verify a design already completed or to study a structure that had failed. Computational demands of practical finite element analysis are so extensive that computer implementation is mandatory.

1.7.2. Steps followed

Elements are connected at points called nodes and the assemblage of elements is called a finite element structure. The particular arrangement of elements is called a mesh. How the finite element method works can be summarised in the following general terms. Regardless of the approach used to find the element properties, the solution of a continuum problem by the FEM always follows an orderly step-by-step process [Reddy (1993)].
1. **Discretizing the continuum:** The first step is to divide the continuum or solution region into elements. A variety of element shapes may be used, and different element shapes may be employed in the same solution region.

2. **Selection of interpolation functions:** The next step is to assign nodes to each element and then choose the interpolation function to represent the variation of the field variable over the element.

3. **Finding the element properties:** Once the finite element model has been established, the matrix equations expressing the properties of the individual elements can be determined using any of the following approach.
   - Direct approach
   - Variational approach
   - Weighted residuals approach

4. **Assembling of the element properties to obtain the system equations:** By combining the matrix equations expressing the behaviour of the elements and forming the matrix equations expressing the behaviour of the entire system, the system equations are formed.

5. **Imposing the boundary conditions:** Before the system equations are ready for solution they must be modified to account for the boundary conditions of the problem.

6. **Solving the system equations:** The assembly process gives a set of simultaneous equations and by solving them, the unknown nodal values of the problem are obtained.

7. **Making additional computations if desired:** Many times the solution of the system equations can be used to calculate other important parameters.
For all FE analysis, whether linear or nonlinear, the FE method is an approximation to reality, the success of which is dependent on

- The ‘quality’ of the FE model, i.e., geometrical accuracy.
- The discretisation process (choice, creation, and distribution of the mesh)
- Material properties and related assumed behaviour
- Representation of loadings and boundary conditions
- The solution process itself (solution method, convergence criteria, etc.)

1.7.3. Advantages

FEA has several advantages over most other numerical analysis methods, including versatility and physical appeal, which are listed below [Cook et al (2002)].

- FEA is applicable to any field problem.
- There is no geometric restriction.
- Boundary conditions and loading are not restricted
- Material properties are not restricted to isotropy and may change from one element to another or even within an element.
- Components that have different behaviours, and different mathematical descriptions, can be combined.
- An FE structure closely resembles the actual body or region to be analyzed and the approximation is easily improved by grading the mesh.

1.7.4 Disadvantages

Some disadvantages may be mentioned as well:

- It is fairly complicated, making it time-consuming and expensive to use.
- It is possible to use finite element analysis programs while having little knowledge of the analysis method or the problem to which it is applied.
- Finite element analysis carried out without sufficient knowledge may lead to results that are worthless and some critics say that most finite element analysis results are worthless [Cook et al (2002)].
1.8. **TYPES OF BUCKLING ANALYSIS**

Two types of buckling analysis exist for a thin shell structures, and are as under:

1. Linear or bifurcation buckling analysis and
2. Nonlinear or collapse buckling analysis

### 1.8.1. Linear or bifurcation buckling analysis

Linear buckling (also called as eigen value buckling or Bifurcation buckling) analysis predicts the theoretical buckling strength of an ideal elastic structure. For elastic or linear buckling analysis, it is assumed that there is no yielding of the structure and the direction of the applied forces does not change. In the case of an ideal plate structure, as the axial load is increased, the lateral (modal) displacement remains zero until the attainment of the critical buckling load. If the axial load versus lateral displacement is plotted, the resulting line / curve will lie along the load axis up to $P = P_{cr}$ as shown in Figure 1.9. This is called the fundamental path. At the critical buckling load, this path bifurcates into a secondary path as shown in Figure 1.9. The secondary path reflects the ability of the plate to carry loads higher than the elastic critical load because of its positive post buckling behaviour. Therefore, elastic buckling of a plate need not be considered as collapse.

![Figure 1.9. Linear (eigen value) buckling curve](image-url)
This analysis involves calculating the point at which the primary load deflection path is bifurcated by a secondary load deflection path. ANSYS finite element software package is used to determine the buckling strength of the perfect thin plate through eigen buckling analysis. In eigen buckling analysis, imperfections and nonlinearities cannot be included. Sub-space iteration scheme can be used to extract the load factor or eigen value. In this buckling analysis, we solve for the eigen values that are scale factors that multiply the applied load in order to produce the critical buckling load. In general only the lowest buckling load is of interest, since the structure will fail before reaching any of the higher-order buckling loads. Therefore usually, only the lowest eigen value needs to be computed.

However, imperfections and nonlinearities prevent most real-world structures from achieving their theoretical elastic buckling strength. For a more accurate approach to predicting instability a nonlinear buckling analysis is performed.

1.8.2. Nonlinear or collapse buckling analysis

This is a more accurate approach since this FE analysis has capability of analyzing the actual structures with imperfections. This approach is highly recommended for design or evaluation of actual structures. This technique employs a non-linear structural analysis with gradually increasing loads to seek the load level at which the structure become unstable. Using this technique, features such as initial imperfections, plastic behaviour etc., can be included in the model. In this analysis, both geometrical and material nonlinearities are utilized. A shell is said to behave nonlinearly if the deflection at any point is not proportional to the magnitude of the applied load [Budiansky (1968) and Brush and Almroth (1975)]. There are two types of nonlinearities: geometric and material. The geometric nonlinearity is the result of nonlinear strain-displacement relations, and the material nonlinearity is the result of nonlinear stress-strain relations. The material non-linearities can also be defined with different work hardening behaviours [Avner (2001)]. Most of the work done on nonlinear shells has taken into account the geometric nonlinearity because traditional engineering materials, such as steel and stainless steel, behave linearly only when the principal strains remain small.
A full incremental nonlinear static stress analysis is used, taking initial displacement (imperfections) matrix into account and applying displacement loading incrementally. In order to find the maximum load carrying capacity (i.e., ultimate collapse load) of the structure accurately, Snap through approach of the non-linear analysis has been followed.

1.9. INTRODUCTION TO LINEAR BUCKLING ANALYSIS

Usually, linear bifurcation buckling is performed in two steps [ANSYS user manual]. In the first step, the structure is loaded by an arbitrary reference level of external load \{R\}_ref and linear static analysis is carried out to determine element stresses such as membrane stresses and from these stresses, stress stiffness matrix \[K_\sigma\]_ref (also called as geometric stiffness matrix) for the given reference load \{R\}_ref load is determined. For some other load level, the stress stiffness matrix will be obtained such that,

\[K_\sigma = \lambda \cdot [K_\sigma]_ref\]  

i.e., \[K_\sigma = \lambda \cdot [K_\sigma]_ref\]  

when \{R\} = \lambda \cdot \{R\}_ref

where \lambda is a scalar multiplier called as buckling load factor (the ratio between bifurcation buckling load and applied load).

The \lambda multiplication with all loads \(R_i\) in \{R\}_ref, results in multiplication of intensity of the stress field by \lambda, but does not alter the distribution of stresses. Because the problem is assumed as linear, the conventional stiffness matrix \[K\] is unchanged by loading. Therefore, the equation of equilibrium just before bifurcation can be written as,

\[\left([K]+\lambda [K_\sigma]_ref\right)\{D\}_ref = \lambda [K_\sigma]_ref \{D\}_ref\]  

Let the buckling displacements \{\delta D\} take place relative to displacements \{D\}_ref of the reference configuration. Because external loads do not change at bifurcation point, the equation of equilibrium just after bifurcation can be written as,
\[
\begin{align*}
\left[ [K] + \lambda_{cr} [K_{\sigma}]_{\text{ref}} \right] \{ \delta D \} &= \lambda_{cr} \{ R \}_{\text{ref}} \\
\text{(1.4)}
\end{align*}
\]

Subtraction of the Eq. (1.3) from the Eq. (1.4) yields

\[
\begin{align*}
\left[ [K] + \lambda_{cr} [K_{\sigma}]_{\text{ref}} \right] \{ \delta D \} &= \{ 0 \} \\
\text{(1.5)}
\end{align*}
\]

\[
\begin{align*}
[K_{\text{net}}] \{ \delta D \} &= \{ 0 \} \\
\text{(1.6)}
\end{align*}
\]

The above Eq. (1.6) is an eigen value problem in which \([K_{\text{net}}]\) is singular and has zero determinants. The physical meaning of the above Eq. (1.6) can be interpreted as the stresses are critical; the net stiffness reduces to zero with respect to the buckling mode displacements \(\{ \delta D \}\). The smallest root \(\lambda_{cr}\) of the above eigen value problem defines the smallest level of external load for which there is bifurcation, i.e.,

\[
\{ R \}_{cr} = \lambda_{cr} \{ R \}_{\text{ref}} \\
\text{(1.7)}
\]

The eigen vector \(\{ \delta D \}\) associated with \(\lambda_{cr}\) is the critical mode. Since, the magnitude of \(\{ \delta D \}\) is indeterminate in linear buckling problem it defines the shape not the amplitude. In the second step the above eigen value problem is solved using either ‘Subspace iteration’ or the ‘Block Lanczos’ techniques to extract the eigen values and eigen vectors. This bifurcation buckling analysis does not account for initial imperfections and also provides no information about post-buckling behavior.

1.10. INTRODUCTION TO NONLINEAR BUCKLING ANALYSIS

When a simple spring of stiffness \(K\) (N/m), is loaded at the free end by a force \(F\) (N), using a simple Hooke’s linear relationship, the force applied and the associated deflection can be related by \(F = Ku\). The deflection \((u)\) can be easily calculated by dividing the force applied by the stiffness coefficient. But, practical applications of such linear systems are somewhat more complex, and are solved by means of ‘matrix’ computational methods. The force and displacement quantities become vectors.

However, the underlying principle is basically the same. Knowing the force or in some cases, the prescribed displacement and the stiffness of the structure, allows
the associated unknown to be calculated. Such structural calculations are of course, valid only where a linear relationship between force and displacement can be assumed to exist.

In reality, all forms of structural mechanics exhibit nonlinear behaviour, whereby the stiffness of a structure progressively changes under increased loading, as a result of material changes, and/or geometric and contact effects. The end result is that the force applied and the displacements observed are no longer linearly proportional. The simulation of such behaviour characterizes the nonlinear analysis and can be summarized by the equation $F \neq Ku$. Both for mathematical and computational convenience, initial FE calculations can be done assuming a linear structural response is indeed a good approximation, especially when the displacements are relatively small (i.e., relative to the structural dimensions) and the material behaviour is linearly elastic. Now a days, it is essential and also it is possible to analyze the real structural systems or designs more closely to the physical reality by including nonlinear effects in their calculations. For example buckling of thin shell structures, automotive crash structures, and many more, are completely reliant on nonlinear behaviour in order to function as intended. Hence, it can be concluded that the assumption of linearity is no longer an acceptable approximation.

As described above, all physical processes have some degree of nonlinearities. For example, pressing an expanded balloon to create a dimple (the stiffness progressively increases, i.e., the rate of deflection decreases as the squeezing force applied increases), or flexing a paper-clip until a permanent deformation is achieved (the material has undergone a permanent change in its characteristic response). These cases and many other practical applications, exhibit either large deformations, and/or inelastic material behavior.

1.10.1. Characteristics of nonlinearities

In general, there are three major, and most common, sources of nonlinear structural behaviour. They are geometric, material and boundary condition nonlinearities.
a) Geometric nonlinearities

Geometric nonlinearities refer to the nonlinearities in the structure or component due to the changing geometry as it deflects, (i.e., the stiffness \([K]\) is a function of the displacements \([u]\).) in turn causing the change in stiffness. Here, the strains are obtained from the displacements via a nonlinear differential operator. This geometric nonlinearity accounts for phenomenon such as the stiffening of a loaded clamped plate, and buckling or ‘snap-through’ behaviour in slender structures or components. There are four types of structural nonlinearities namely, large strain, large rotation, stress stiffening, and spin softening.

Large strain assumes that the strains are no longer infinitesimal (they are finite). Shape changes (e.g. area, thickness, etc.) are also taken into account. Deflections and rotations may be arbitrarily large. Large rotation assumes that the rotations are large but the mechanical strains (those that cause stresses) are evaluated using linearised expressions. The structure is assumed not to change shape except for rigid body motions. Stress stiffening assumes that both strains and rotations are small. First order approximation to the rotations is used to capture some nonlinear rotation effects. Spin softening also assumes that both strains and rotations are small. This option accounts for the radial motion of a body's structural mass as it is subjected to an angular velocity. Hence it is a type of large deflection but small rotation approximation.

b) Material nonlinearities

Material Nonlinearity refers to the ability for a material to exhibit a nonlinear stress-strain (constitutive) response which may be dependent on stresses, strains and/or displacements. In FE analysis, many common engineering materials are assumed to have an elastic initial response and an inelastic (often plastic) response once the stress reaches yield limit. Plastically deformed materials exhibit different stiffness characteristics to their original elastic state and show permanent deformations if the loading is removed. There are a large number of theoretical material models which can describe both simple and complex elasto-plastic behaviour and these material behavior can be broadly divided into two classes:
i) **Time-independent behaviour**: In this case there is no subsequent straining should loadings be maintained over a long period of time. This applies to many ductile metals, rubber and elastomers.

ii) **Time-dependent behavior**: Often referred to as ‘visco’ effects (visco-elasticity, or visco-plasticity) or more simply ‘creep’, time dependent materials are assumed to develop additional straining over long periods of time. Elevated temperature conditions also contribute to time-dependent effects. Visco-plasticity is typically applicable to high-temperature applications and applied for materials like elastomers, glass, and plastics.

iii) **Yield criteria**: The yield stress is a measured stress value that marks the onset of plastic deformation. Its magnitude is usually obtained from a uniaxial test. However, since stresses in a structure are multi-axial, this uniaxial value is typically used in a multi-axial measure of yielding called the ‘yield condition’ or ‘yield criterion’. The most widely used yield condition is the von Mises criterion in which yielding is assumed to occur when the level of effective stress (often referred to as ‘equivalent’ or von ‘Mises stresses) reaches the yield value as defined from a uniaxial test. This yield condition agrees fairly well with the observed behavior of many ductile metals such as low carbon steels and aluminum, and is widely used in materially nonlinear FE analysis. Another commonly used yield condition is the Tresca condition, in which yielding occurs when the maximum shear stress reaches the value it has when yielding occurs in the uniaxial tensile test.

iv) **Work hardening**: In a uniaxial test, the work hardening slope (the post-yielding stiffness) is defined as the slope of the stress to plastic strain curve. It relates the incremental stress produced to the incremental plastic strain applied and dictates the conditions of post-yield material behaviour. An ‘isotropic’ hardening rule assumes that the center of the yield surface remains stationary in the stress space, but the size (radius) of the yield surface expands due to strain hardening. This material property is considered suitable for problems in which the plastic
straining far exceeds the incipient yield state. It is therefore widely used for simulations of the manufacturing processes and large-motion dynamic problems. The ‘kinematic’ hardening rule assumes that the von Mises yield surface does not change in size or shape, but the center of the yield surface shifts in stress space. Straining in one direction reduces the yield stress in the opposite direction. It is therefore used in cases where it is important to model behaviour such as the Bauschinger effect.

c) **Boundary condition nonlinearities**

In this case, the material and strains remain linear in behavior. The nonlinear behavior comes from changing of boundary. When FE analysis was performed on either highly flexible components, or structural assemblies comprising multiple components, progressive displacement gives rise to the possibility of either self or component-to-component contact. Since the detection of contact (or separation) is dependent on a continual monitoring of an updated geometry configuration, contact is by nature a type of geometric nonlinearity. However it is often referred to as boundary condition nonlinearity since the change in contact conditions act in a similar way to changes in boundary conditions such as loads or constraints. In boundary condition nonlinearity the stiffness of the structure or assembly may change when two or more parts either in contact or separate from initial contact. During contact, mechanical loads, and perhaps thermal or other physical entities, are transmitted across the area of contact. If friction is present, shear forces are also considered.

1.10.2. **Sources of nonlinearities**

In nonlinear FEA, following important nonlinear effects are considered.

a) **Large strains**

Most metallic materials (in which small strains effects are considered) are no longer structurally useful when the strain exceeds one or two percent. However, some
materials, notably rubbers, other elastomers, and plastics, can be strained to hundreds of a percent. In such applications it is required to consider large strains effects in their numerical analysis.

\textit{b) Nonlinear strain-displacement (compatibility) relationships}

In applications involving large structural rotations (even for small strains) nonlinear strain-displacement relationships should be considered. In such cases the changes in the deformed shape can no longer be ignored. The mechanics of buckling, rubber analysis, metal forming, and many other common applications, require that the often ignored higher derivative terms in the strain displacement relationship are considered in the formulation.

\textit{c) Nonlinear stress-strain (constitutive) relationships}

The nonlinear constitutive relationships relate to the progressive change in a material response depending on the amount of strain (i.e., indirectly the displacement) which it experiences. The common example is plasticity that a material typically experience after yield point which significantly reduces the load carrying capacity. If the material is unloaded the post-yield strain level produces a permanent material deformation. On continuation of loading, plasticity may continue to develop until displacements effectively become infinitely large for an infinitely small increase in load. At this stage the material is assumed to be ‘fully’ plastic and has lost all structural engineering performance. In some cases post-yield hardening may occur as strains further increase. Metals, plastics and many other materials exhibit different forms of elasto-plastic behaviour under a variety of different loading conditions. Many other forms of material nonlinearity also exist, of which cracking/fracture, crushing (concretes/foams), and de-lamination (laminated composites) are perhaps the most commonly encountered.

1.10.3. Solution procedures

Nonlinear systems of equations are most commonly solved using iterative incremental techniques where small incremental changes in load are found by
imposing small incremental changes in displacement on the structure. The resulting solutions are used to plot a curve in space, which is referred to as the equilibrium path for the structure.

While solving the system of simultaneous equations in nonlinear analysis, using the Newton-Raphson procedure, the tangent stiffness $K_T$ (the relationship between the incremental load and the associated displacement) is used and this comprises of three components namely material stiffness, initial stress stiffness and geometric stiffness. The material stiffness may be the same elastic stiffness as used in linear FE solutions. The initial stress stiffness term represents the resistance to load caused by realigning the current internal stresses when displacements occur. The final component, the geometric stiffness, represents the additional stiffness due to any nonlinearity in the strain-displacement relationship.

**a) Updated equilibrium configuration**

Equilibrium is one of the important principles of the nonlinear FE method, and that ensures externally applied forces and internally generated states of stress are balanced. The overall result of nonlinearity (from whatever source of origin) is that the force-displacement relationship requires continual updating in order to maintain equilibrium, and also for the physical validity of the simulation. This numerical equilibrium is typically maintained by solving nonlinear equations with an ‘incremental-iterative’ approach.

**b) Incremental-iterative solution procedures**

The method of solution of the simultaneous equations which describes the behaviour of any application is one of the important considerations, especially when the application is nonlinear in nature. In a simple linear analysis, the loads are applied and the displacements are calculated from a relatively simple inversion of the stiffness matrix. In nonlinear analysis the non-proportionality between the applied load and the resulting displacements (the result of the inherent nonlinearity in the structural stiffness) is accounted for by applying the load in a series of steps or ‘increments’. Early nonlinear analysis used a purely incremental approach, but depending on the
size of the increments used and the degree of nonlinearity encountered such techniques often diverged from the true structural behaviour. This approach may be used as a first trial in order to get acquainted with the problem and determine appropriate parameters such as the step (or time increment) size, number of steps, equilibrium checks, tolerances for convergence, and other control parameters.

In the advanced solution techniques, within each load increment the loss of equilibrium between individual load increments is corrected using an iterative technique as shown in Figure 1.10. Such methods are therefore referred to as ‘incremental-iterative’ (or Newton-Raphson) methods and widely used in nonlinear analysis. The construction and inversion of the stiffness matrix is one of the major computational overheads in performing a nonlinear analysis. This is especially true for models with a large number of degrees of freedom, and/or a large number of loading increments.

![Figure 1.10. Incremental-iterative or standard Newton-Raphson method](image1.png)

![Figure 1.11. Modified Newton-Raphson method](image2.png)

In the standard Newton-Raphson method, the stiffness matrix is reformed for each iteration of each increment in the process. Because of continually current nature of the structural stiffness, standard Newton-Raphson methods convergence is usually fairly rapid (few correcting iterations are required). However, the computational expense of each iteration may be considerable. For this reason ‘modified’ Newton-Raphson methods are often used, in which the stiffness matrix is reformed less frequently (perhaps just on the first or second iteration of each increment where the stiffness changes most significantly). Modified Newton-Raphson methods as
described above generally converge less rapidly (more iterations are required) as shown in Figure 1.11, but a net gain in computational efficiency is achieved in the less frequent formation and inversion of the stiffness matrix.

There are other alternative solution procedures available to solve the nonlinear equations, namely:

- Strain Correction Methods
- Secant or Conjugate-Gradient Methods
- Direct Substitution Methods
- Quasi-Newton Methods

c) **Arc Length Method**

The major difficulty in solving the system of equation using either Newton-Raphson or modified Newton-Raphson technique is due to the singularity of the tangent stiffness matrix $K_T$ when the structure reaches its stability limit. In addition, snap-through and snap-back buckling phenomena pose some of the most difficult problems in nonlinear structural analysis. Riks (1979) and Crisfield (1981) introduced constraint methods which guides the solution to follow a certain path. A method called “arc length automatic stepping” as shown in Figure 1.12, is a scheme which can be applied to overcome the problem of stiffness singularity and post-buckling. The arc length methods are very similar to the Newton-Raphson method except that the applied load increment becomes an additional unknown. In this method the solution is made to converge along an arc as shown in Figure 1.12. The goal of arc length procedure is to control of iteration in the numerical solution of complex nonlinear problems. In this method, the incremental displacement length for each successive iteration is constrained by the length of the previous iteration. Correspondingly, the load is adjusted in order to satisfy the global equilibrium requirement of the system.
1.10.4. Description of geometric state of the structure

As previously described, in solving this type of nonlinear problems, the load is increased in small increments and the incremental displacement is found using the current (or near current) approximation to the tangent stiffness. However, in geometrically nonlinear (or large displacement) analysis, there are a number of different ways in which the geometric state of the structure can be described and some of the important descriptions are given below.

a) Total Lagrangian description

In a ‘Total Lagrangian’ method all calculations, at each stage of the incremental loading history, are always referred to the original (undeformed) geometry. For example, the calculation of items such as stress always utilizes the original representative area in the force/area calculation. Total Lagrangian methods are therefore generally suitable for applications which exhibit large displacements and large rotations, but where strains are generally small.

b) Updated Lagrangian description

As the name suggests, an ‘Updated Lagrangian’ method always uses the current (deformed) configuration of the structure for calculation. This requires the FE
mesh coordinates to be updated at each increment in order to form a new ‘reference geometry’. For this reason an Updated Lagrangian approach is more suitable to applications such as metal forming, which exhibit large inelastic strains.

c) Eulerian description

An Eulerian description has a major difference in approach. In the Eulerian method it is assumed that as the material of the structure deforms its motion is described relative to a fixed spatial geometry that is the material flows through a fixed frame of geometric reference. This approach has major advantages when describing materials which essentially flow, such as structural forming, extrusion or classical fluid dynamics. In some cases mixed formulations can be used. For example in the structure-fluid coupling, is achieved by combining a Lagrangian type formulation for the structural aspects of the model, with a Eulerian approach to describe the motion of the fluid components. Such descriptions are of course purely mathematical conveniences, but are highly useful for coupled applications such as the gas dynamics of airbags in passenger safety, aero-engine bird strike, helicopter sea-ditching etc.

1.10.5. Convergence criteria

General non-linear FE program will continue to do equilibrium iterations until the convergence criteria are satisfied or until the maximum number of equilibrium equations is reached. There are two types of convergence criteria namely force and displacement criteria. The displacement convergence checking is based on checking the change in deflections (Δu) between the current (i) and the previous (i-1) iterations: Δu = u_i - u_{i-1}. The force [and, when rotational degrees of freedom (DoF) are active, moment] convergence checking is done by comparing the square root sum of the squares (SRSS) of the force imbalances of all DoF against the SRSS of the applied loads and tolerance imposed. The loose tolerance values on convergence checking may lead to inaccurate results, whereas tight tolerance values may be uneconomical from the computational point of view. In general, the use of only one criterion (specially the force one) may not be adequate for general applications. For example in some applications of elasto-plastic problems with zero or very small strain hardening
modulus, the force criterion may be easily satisfied while the displacement one is in significant error. Usually, 0.5% and 5% tolerance setting will be used for force convergence and displacement convergence checking.

1.10.6. Stress stiffening

The out-of-plane stiffness (bending) of a structure can be significantly affected by the state (compression/tension) of in-plane stress (membrane stress) in that structure. This coupling between in-plane stress and lateral stiffness, known as stress stiffening, which is more pronounced in thin, highly stressed structures, such as cables or membranes. For example, a drum head membrane gains lateral stiffness as it is tightened. If the membrane stress is compressive, the resistance offered by stress stiffening, to bending deformation is reduced and also if the membrane stress is tensile, resistance to bending deformation is increased. In case of solving the problem of large strain and large deflection using nonlinear analysis the stress stiffening should be included as initial stress effects while the nonlinear analysis procedure is employed.

1.11. MATERIAL MODELING

The phenomenon of buckling can be categorized (by plasticity) into three classes, namely elastic buckling, elastic–plastic buckling and plastic buckling where the last two are called inelastic buckling. Elastic buckling is only observed in the elastic regime. On the other hand elastic–plastic buckling occurs after a local region inside the plate deforms plastically. Plastic buckling refers to buckling that occurs in the regime of gross yielding, i.e., after the plate has yielded over large areas (Paik and Thayamballi, 2003). These three classes of buckling are further explained as below.

1. **Elastic buckling**: It is a process that initiates at the critical states with elastic material properties. Thus, instability occurs before plasticity: when the structure reaches plastic deformations it already experienced buckling. This occurs in most thin-walled shells, such as tanks.
2. **Plastic buckling:** It is a process that initiates with plastic deformations. Thus, plasticity occurs before instability: when the structure reaches a buckling load it already had plastic deformations. This occurs in thick shells.

3. **Elastic-plastic buckling:** This occurs when plasticity and instability occur almost at the same load level. This occurs in moderately thin shells.

There are number of mathematical formations available to describe the physical behaviours of a material. In this section elasto-plastic behaviour of the material is considered to identify the appropriate form in which isotropic metals like steel, aluminium, etc., can be defined for FE analysis.

### 1.11.1. Idealized uniaxial stress-strain curves

Three idealized stress-strain curves for prismatic metal bars subjected to uniaxial tension are namely rigid-perfectly plastic, elastic-perfectly plastic, and elastic-plastic with strain hardening behaviors and are shown in Figure 1.13 (a), (b), and (c), respectively. The yield strength of the material is denoted by ‘$\sigma_y$’ in the plots.

![Figure 1.13. Idealized stress-strain curves for different material behaviours](image)

The rigid-perfectly plastic idealization neglects elastic strains and hardening. The elastic-perfectly plastic curve includes elastic strains but neglects hardening. The elastic, strain-hardening curve includes elastic strains and assumes linear hardening. When large deformations are prevented, say, by a surrounding elastic material, then plastic deformation is contained locally. For contained plastic deformation, neglecting strain hardening, or work-hardening, is a reasonable assumption. Large deformations
by cold-working occur in metal-forming processes such as drawing, rolling, and extrusion. Cold-working involves hardening and the plastic deformations in these processes are much larger than elastic deformations, so that neglecting elastic deformation is a reasonable assumption.

For the structural elements made-up of carbon steel or structural steel, the stress strain curve can be modeled as elastic and perfectly plastic curve which is a reasonable assumption, since in these materials strain hardening will be pronounced only after certain plastic yield as shown in Figure 1.14 (a) [Dieter (1988)]. In case of aluminium, it is reasonable to assume stress-strain behavior as shown in Figure 1.14. (b) with isotropic hardening [Donald Lesuer. (2000)].

![Stress-strain curves](image)

(a) Carbon steel  
(b) Al 2024-T3 alloy

Figure 1.14. Typical stress-strain curves of carbon steel and Al 2024-T3 alloy

1.11.2. Elastic-plastic stress analysis

a) Fundamental assumptions

Following assumptions apply to material model formulations [Cook et al. (2002)]:

1. The material is treated as a continuous medium or a continuum.
2. The material is isotropic, with its properties independent of direction.
3. The material has no “memory” such that the effect on the material in previous events does not impact the current event.
b) *Difference between elastic and plastic deformation of solids*

Some of the fundamental differences between elastic and plastic material behavior [Callister (1995)] are

**i) Elastic deformation:**
1. Very small deformation with the strain up to about 0.1%
2. Usually a linear relationship between the stress and strain.
3. Completely recoverable strain or deformation after the applied load is removed.

**ii) Plastic deformation:**
1. Larger deformation.
2. Nonlinear relationship between the stress and strain.
3. Results in permanent deformation after removal of applied load.
4. No volumetric change in the solid during plastic deformation, often modeled with Poisson’s ratio of 0.5. Deformation is caused by shear actions on the material. Only shape changes can be observed.
5. The total strain, $\varepsilon_T$, is the sum of the elastic component $\varepsilon_e$ and the plastic component $\varepsilon_P$ as shown in Figure 1.15.

![Figure 1.15](image_url)  
*Figure 1.15. Uniaxial elastic-plastic loading and elastic unloading of a material [Callister (1995)]*

Where there is yielding, not only the load, deformations and stresses are also non-linearly related and they are also load history dependent. The plasticity refers to
deformations that are not recovered if the loads are removed. In case of static buckling
problems, the plasticity is considered as time independent i.e., creep effect and stress
rate are not considered.

Three essential ingredients of elastic-plastic analysis [Cook et al. (2002)] are

- The existence of an initial yield surface which defines the elastic limit of
  the material in a multi-axial state of stresses. (i.e., a yield criterion)
- The hardening rule describes the progress in stress state, subsequent to
  yield surface.
- The flow rule which defines the direction of the incremental plastic strain
  vector in stress space.

The yield strength of a metal is usually measured in the tension tests, which
are of uniaxial state of stress and there is no theoretical way to correlate yielding in a
three-dimensional stress state with yielding in the uniaxial tensile test. The yield
condition now can only be defined by the “yield criterion” [Callister (1995)].

c) von Mises yield criterion

Prior to any yielding, many metals display almost linear elastic response, so
that the stresses can be calculated by knowing elastic constants. For any element of
material in a given state of stress (following some stress history) there exists a region
in the stress space such that the behavior is elastic and path independent if the stress
point lies within this region. The elastic region is bounded by a yield surface in the
stress space, as shown in Figure 1.16.

Most of the common metals obey the von Mises yield criterion which was
derived from the distortion energy theory. Let σ_e denote the von Mises effective stress
defined by

$$
σ_e = \frac{1}{\sqrt{2}} \left[ (σ_x - σ_y)^2 + (σ_y - σ_z)^2 + (σ_z - σ_x)^2 + 6(τ_{xy}^2 + τ_{yz}^2 + τ_{zx}^2) \right]^{1/2}
$$

where σ_x, σ_y, τ_{zx} are the Cartesian stress components at a point in the material.
Figure 1.16. Two-dimensional von Mises yield surface [Callister (1995)]

The von Mises stress criterion states that yielding initiates in a three-dimensional state of stress when the effective stress equals the yield strength of the material determined from the uniaxial tensile test. Expressed mathematically the criterion is \( \sigma_e = \sigma_{\text{yield}} \) at the initiation of yielding.

For the uniaxial state of stress where \( \sigma_x \neq 0 \) and all other stresses components are equal to zero, von Mises criterion predicts yield initiation when \( \sigma_x = \pm \sigma_{\text{yield}} \). For the state of pure shear where \( \tau_{xy} \neq 0 \) and all other stress components are equal to zero, von Mises criterion predicts the initiation of yielding when \( \tau_{xy} = \pm \sigma_{\text{yield}}/(\sqrt{3}) \). Thus, von Mises criterion implies that the yield stress in tension is \( \sqrt{3} \) times the yield stress in shear. This relationship between the yield stresses in tension and pure shear closely approximates many tests of polycrystalline metals [Callister (1995)]. von Mises criterion can be visualized in principal stress space, where the principal stresses are denoted by \( \sigma_1, \sigma_2 \) and \( \sigma_3 \). The criterion plots as right circular cylindrical surface of radius, with the axis of the cylinder equally inclined with respect to the positive principal stress axes. The von Mises stress criterion can be plotted as an ellipse in 2D as shown in Figure 1.16.

Deviatoric stresses play a predominant role in von Mises theory. Any stress state can be represented as the sum of a hydrostatic state and a deviatoric state. The hydrostatic state produces no change of shape. A deviatoric state produces no change of volume. Deviatoric normal stresses are actual normal stresses minus the mean normal stresses \( \sigma_m \) where, \( \sigma_m = (\sigma_x + \sigma_y + \sigma_z)/3 \). Deviatoric shear stresses are the same as
actual shear stresses. Adopting the symbol \( s \) for deviatoric stresses, the deviatoric normal stresses can be written as

\[
\{S_{\sigma}\} = \begin{bmatrix}
s_x \\
s_y \\
s_z
\end{bmatrix} = \begin{bmatrix}
\sigma_x - \sigma_m \\
\sigma_y - \sigma_m \\
\sigma_z - \sigma_m
\end{bmatrix} = \frac{1}{3} \begin{bmatrix}
2\sigma_x - \sigma_y - \sigma_z \\
2\sigma_y - \sigma_z - \sigma_x \\
2\sigma_z - \sigma_x - \sigma_y
\end{bmatrix}
\]  \hspace{1cm} (1.9)

and also \( s_x + s_y + s_z = 0 \)  \hspace{1cm} (1.10)

Similarly the deviatoric shear stresses can be written as

\[
\{S_{\tau}\} = \begin{bmatrix}
s_{xy} \\
s_{yz} \\
s_{zx}
\end{bmatrix} = \begin{bmatrix}
\tau_{xy} \\
\tau_{yz} \\
\tau_{zx}
\end{bmatrix}
\]  \hspace{1cm} (1.11)

von Mises effective stress \( \sigma_e \) can also be written in terms of deviatoric stresses as

\[
\sigma_e = \sqrt{\frac{3}{2} \left[ s_x^2 + s_y^2 + s_z^2 + 2(s_{xy}^2 + s_{yz}^2 + s_{zx}^2) \right]} \hspace{1cm} (1.12)
\]

d) \hspace{1cm} Strain hardening

Callister (1995) defines strain hardening as the increase in hardness or yield strength of a ductile metal as it is plastically deformed. Slip begins at an imperfection in the lattice, e.g., along a plane separating two regions, one having one more atom per row than the other. Because slip does not occur simultaneously along every atomic plane, the deformation appears discontinuous on the microscopic level of the crystal grains. The overall effect, however, is plastic shear along certain slip planes. As the deformation continues, the locking of dislocations takes place, resulting in strain hardening.

Initial plastic yielding takes place when a material is loaded beyond its elastic limit. Theoretically, the material starts to “flow” without any additional load as depicted in Figure 1.13 (b). However, in reality most materials retain some of their original stiffness after yielding. Additional loading is required to further plastically deform the material as shown in Figure 1.13 (c) and also the material becomes
“harder” after some plastic deformation as a higher applied load is required to cause the same material to deform plastically again after the completion of one previous loading cycle [Callister (1995)]. Two types of strain-hardening schemes, namely (a) isotropic hardening and (b) kinematic hardening, are typically used in finite element analysis.

i) **Isotropic hardening:**

The Figure 1.17 illustrates the principle of isotropic hardening of a material. First the material is loaded beyond its initial yield strength $\sigma_y$ to an instantaneous strain $\varepsilon_1$ and then unloaded upon reaching point A. A permanent strain $\varepsilon_2$ is introduced in the material after unloading. If the solid is loaded again, the material is found to yield at a higher strength $\sigma_y'$ which coincides with the stress at $\sigma_A$ the last load point. The biaxial stress states for the isotropic strain-hardening behavior can be graphically represented by the uniform expansion of the initial yield surface as shown in Figure 1.18. The yield surface maintains its shape and does not translate, while its size increase is controlled by stress state $\sigma_0$ which the maximum plastic stress state reached on the previous load cycle.

![Figure 1.17. Paths for loading, unloading and reloading of an isotropic hardening solid [Callister (1995)]](image1)

![Figure 1.18. Expansion of yield surface due to plastic yield of an isotropic hardening solid [Callister (1995)]](image2)

ii) **Kinematic hardening:**

A material which exhibits kinematic hardening has the characteristic that when the material is plastically deformed in tension followed by loading in the reverse direction, the compressive yield strength in reverse loading is reduced by the same
amount as that of excess of applied stress over the tensile yield strength during the initial loading. The lowering of the compression yield following a first loading in tension is called the Bauschinger effect.

![Figure 1.19. Paths for a uniaxially loaded bar under kinematic hardening plastic deformation [Callister (1995)]](image1)

![Figure 1.20. Translation of the yield surface with kinematic hardening under biaxial rule loading condition [Callister (1995)]](image2)

But, the material with isotropic hardening character has same magnitude of yield strengths both in tension and in compression as in Figure 1.19 (Load history path OABC’O represents the behaviour of material with isotropic hardening effect). Thus, the isotropic hardening model cannot predict the Bauschinger effect, but the kinematic model can predict it. Kinematic hardening results in the strain hysteresis observed after a complete tension-compression load cycle, as shown in Figure 1.19. In the case the multiaxial loading situation, the size and shape of the yield surface does not change, but it merely translates in stress space as is shown in Figure 1.20. The translation is in a direction of the plastic-deformation-increment vector.

e) **Flow Rule**

The flow rule relates the state of stress \( \{\sigma\} \) to the corresponding six increments of plastic strains \( \{d\varepsilon^p\} \) when the increment of plastic flow occurs. The general accepted flow rule is that plastic strain occurs normal to the yield surface as shown in Figure 1.21 [Zienkiewicz (2002)] and is given by

\[
\{d\varepsilon^p\} = \left[ \frac{\partial F}{\partial \sigma} \right] d\lambda
\]  

(1.13)
In this $d\lambda$ is proportionality constant and is a scalar, also known as plastic multiplier. This rule is also known as normality principle because the above relationship can be interpreted as requiring the normality of the plastic strain increment ‘vector’ to the yield surface in the space of n-stress dimensions.

The flow rule is also stated in terms of function $Q$ (in case of associated plasticity where $Q = F$) which has unit of stress and is called as ‘plastic potential’

$$\{d\varepsilon^p\} = \left[\frac{\partial Q}{\partial \sigma}\right] d\lambda$$

(1.14)

or for any component of $n$

$$\{d\varepsilon_n^p\} = \left[\frac{\partial Q}{\partial \sigma_n}\right] d\lambda$$

(1.15)

1.11.3. Introduction to FE elastic-plastic analysis

In the nonlinear FE analysis, both incremental and iterative methods are used to model the nonlinear elasticity and elasto-plasticity. In addition, two modified methods are also employed for elastic plastic analysis: the initial strain method and initial stress method. The most common solution procedure for elastic plastic problems is incremental method utilizing the tangent stiffness concept. The iterative method and other modified methods are occasionally employed.
The element tangent stiffness matrix for FE non-linear analysis is computed from the usual relationship

\[
[K_t] = \int [B]^T [E_{ep}] [B] \, dv 
\]  

(1.16)

where \([B]\) is strain displacement matrix and \([E_{ep}]\) is elastic–plastic matrix. When the material is in elastic condition \([E_{ep}] = [E]\) and when the material is in plastic condition \([E_{ep}]\) should be calculated including plasticity effect. In this section, incremental method to calculate \([E_{ep}]\) including plasticity is discussed.

The three essential ingredients of elastic-plastic analysis are a yield criterion, a flow rule and a hardening rule. The yield criterion relates the state of stress to the onset of yielding. The flow rule relates the state of stress \(\{\sigma\}\) to the corresponding six increments of the plastic strain \(\{d\varepsilon^p\}\) when an increment of plastic flow occurs. The hardening rule describes how the yield criterion is modified by straining beyond initial yield. Experimental evidence supports the assumption that during plastic deformation essentially no volume change occurs; that is, the material is incompressible [Callister (1995)]. Thus

\[
\varepsilon_x^p + \varepsilon_y^p + \varepsilon_z^p = \varepsilon_1^p + \varepsilon_2^p + \varepsilon_3^p = 0 
\]  

(1.17)

The strain increment during plastic condition is considered as composed of recoverable (elastic) and non-recoverable (plastic) components i.e.,

\[
d\varepsilon = d\varepsilon^e + d\varepsilon^p 
\]  

(1.18)

where the superscript e and p denote elasticity and plasticity respectively. But the stress increments are due to only elastic component. Therefore

\[
d\sigma = E d\varepsilon^e 
\]  

(1.19)

(or)

\[
d\sigma = E (d\varepsilon - d\varepsilon^p) 
\]  

(1.20)

Applying flow rule (Eq. 1.14) in the above Eq. (1.20)

\[
d\sigma = E \left( d\varepsilon - \left( \frac{\partial \varepsilon}{\partial \sigma} \right) d\lambda \right) 
\]  

(1.21)
and \[ \{d\sigma\} = \left[ d\sigma_x \quad d\sigma_y \quad d\sigma_z \quad d\tau_{xy} \quad d\tau_{yz} \quad d\tau_{zx} \right]^T \] (1.22)

Hardening can be modeled as isotropic or kinematic either separately or in combination. Isotropic hardening can be represented by plastic work by unit volume \( (w_p) \) which describes the growth of yield surface.

\[ W_p = \int [\sigma]^T \{de^p\} \] (1.23)

\[ \therefore \ dW_p = [\sigma]^T \{de^p\} \] (1.24)

The kinematic hardening can be represented by a vector called kinematic shift \( \{\alpha\} \) which accounts for translation for yield surface in the stress space.

\[ \{d\alpha\} = [C] \{de^p\} \] (1.25)

and in the integral form \[ \{\alpha\} = \int [C] \{de^p\} \] (1.26)

The diagonal matrix \([C]\) is given by

\[
C = \frac{2}{3} H_p \begin{bmatrix}
1 \\
0 & 1 \\
0 & 0 & 1 & \text{sym} \\
0 & 0 & 0 & \frac{1}{2} \\
0 & 0 & 0 & 0 & \frac{1}{2} \\
0 & 0 & 0 & 0 & 0 & \frac{1}{2}
\end{bmatrix}
\] (1.27)

where \(H_p\) is plastic modulus or strain hardening parameter which relates the incremental stress and plastic strain i.e.,

\[ d\sigma = H_p \ d\varepsilon^p \] (1.28)

\[ H_p = \frac{E_T}{1 - (E_T/E)} \] (1.29)

or \[ E_T = E \left(1 - \frac{E}{E + H_p}\right) \] (1.30)

where \(E_T\) is tangent modulus shown in Figure 1.22.
Figure 1.22. Stiffness of the elastic-plastic material in elastic and plastic regions

If $H_p = 0$; for which $E_T = 0$ then the material is called is elastic-perfectly plastic. If the material has not yet yield or is in unloading then $E_T = E$ and $H_p$ is not used of calculation. The plastic flow takes place at a constant volume and since Poisson’s ratio is 0.5 for plastic flow.

\[ \frac{d}{d\varepsilon_x} + \frac{d}{d\varepsilon_y} + \frac{d}{d\varepsilon_z} = 0 \]  

(1.31)

Hence \[ [1 \ 1 \ 0 \ 0 \ 0] \{\alpha\} = \alpha_x + \alpha_y + \alpha_z = 0 \]  

(1.32)

a) Incremental Stress-strain relationship

During an increment of plastic strain $dF=0$ then

\[ \left[ \frac{\partial F}{\partial \sigma} \right]^T d\sigma + \left[ \frac{\partial F}{\partial \alpha} \right]^T d\sigma + \frac{\partial F}{\partial W_p} dW_p = 0 \]  

(1.33)

Substituting Eqs. (1.21), (1.24) and (1.25) in the above Eq. (1.32) and solving for plastic multiplier $d\lambda$ and the resulting equation in the form of

\[ d\lambda = \left[ P_\lambda \right] \{d\varepsilon\} \]  

(1.34)

where, \( \{P_\lambda\} \) is a row matrix given by

\[ \left[ P_\lambda \right] = \left[ \frac{\partial F}{\partial \sigma} \right]^T E \]  

(1.35)
Similarly, substituting Eqs. (1.21), (1.24) (1.30) and (1.34) in Eq. (1.33) and solving for incremental stresses in terms of incremental strains and it can be written as

$$\{d\sigma\} = [E_{ep}]\{d\varepsilon\}$$  \hspace{1cm} (1.36)

where

$$[E_{ep}] = [E][I] - \left( \frac{\partial Q}{\partial \sigma} \right)[P_\lambda]$$  \hspace{1cm} (1.37)

where [I] is unit matrix.

To determine $\left( \frac{\partial Q}{\partial \sigma} \right)$ and [P_\lambda] the yield surface has to be described. Since most of the metals obey von-Mises criteria, this criterion is used to define the plastic yield surface. For von-Mises theory the plastic multiplier $d\lambda$ is same as the increment of effective plastic strain that corresponds to effective stress $\sigma_e$ thus

$$d\lambda = d\varepsilon^p = \sqrt{\frac{2}{3}} \left( (d\varepsilon_x^p)^2 + (d\varepsilon_y^p)^2 + \frac{1}{2} (d\gamma_{xy}^p)^2 + (d\gamma_{yz}^p)^2 + (d\gamma_{zx}^p)^2 \right)^{\frac{1}{2}}$$  \hspace{1cm} (1.38)

Using von-Mises stress criteria the expression for yield function for isotropic hardening can be written as

$$F = \sigma_e - \sigma_0 = 0$$  \hspace{1cm} (1.39)

where $\sigma_0$ is the largest $\sigma_e$ reached in previous plastic strain. In the case $F<0$ describes elastic condition. The case $F=0$ defines yielding. The case $F>0$ is not physically possible.

Similarly “mixed” hardening rule including both isotropic and kinematic hardening by introducing new variable $\eta$ where $0\leq\eta\leq1$. Thus

$$F = \left[ \frac{3}{2} \left( (\varepsilon_x - \eta \alpha_x)^2 + (\varepsilon_y - \eta \alpha_y)^2 \right) + \frac{1}{2} (\varepsilon_{xy} + (\varepsilon_{yz} - \eta \alpha_{yz})^2 + (\varepsilon_{zx} - \eta \alpha_{zx})^2 \right) \right]^{\frac{1}{2}} - \eta \sigma_y - (1 - \eta) \sigma_0$$  \hspace{1cm} (1.40)

where $\sigma_y$ is initial yield stress obtained from uniaxial test. The translation of yield surface is controlled by kinematic shift vector $\{\alpha\}$,

$$\{\alpha\} = \left[ \alpha_x \alpha_y \alpha_z \alpha_{xy} \alpha_{yz} \alpha_{zx} \right]^T$$  \hspace{1cm} (1.41)
Substituting \( \eta=0 \) in the Eq. (1.40) results in isotropic hardening and substituting \( \eta=1 \) in the Eq. (1.40) results in kinematic hardening. And also it can be proved that

\[
\left\{ \frac{\partial F}{\partial \alpha} \right\} = -\eta \left\{ \frac{\partial F}{\partial \sigma} \right\}
\]  (1.42)

The last term in the equation (1.33) \( \frac{\partial F}{\partial W_p} dW_p \) is replace by \( \frac{\partial F}{\partial \sigma_0} d\sigma_0 \) to account for strain hardening instead of work hardening. \( \sigma_0 \) is the max stress reached in previous loading. Substituting \( \frac{\partial F}{\partial \sigma_0} = -(1-\eta) \) obtained from Eq. (1.40) and Eq. (1.28) and also applying associated plasticity rule \( (F=Q) \) into Eq. (1.34) and on simplifying, we get

\[
[P_h] = \frac{\left[ \frac{\partial F}{\partial \sigma} \right]^T E \left( E + \eta[C] \frac{\partial F}{\partial \sigma} \right) + (1-\eta)H_p}{\left[ \frac{\partial F}{\partial \sigma} \right]^T E + \eta[C] \frac{\partial F}{\partial \sigma} + (1-\eta)H_p}
\]  (1.43)

Differentiating \( F \) with respect to \( \{\sigma\} \) is accomplished as follows

\[
\left\{ \frac{\partial F}{\partial \sigma} \right\} = \frac{1}{2} \left[ \frac{\partial [...]}{\partial \sigma} \right] = \frac{1}{2[\eta \sigma_y + (1-\eta)\sigma_0]} \left[ \frac{\partial [...]}{\partial \sigma} \right]
\]  (1.44)

where [...] represents the expression within square brackets in Eq. (1.40). The substitution of denominator in the later form of the above Eq. (1.44) is allowed because \( F=0 \) during yielding. The partial differential term in the flower brackets in the above Eq. (1.44) is evaluated as follows,

\[
\frac{\partial [...]}{\partial \sigma_x} = 3 \left[ (s_x - \eta \alpha_x) \frac{\partial S_x}{\partial \sigma_x} + (s_y - \eta \alpha_y) \frac{\partial S_y}{\partial \sigma_x} + (s_z - \eta \alpha_z) \frac{\partial S_z}{\partial \sigma_x} \right]
\]

\[
\frac{\partial [...]}{\partial \sigma_y} = 3 \left[ (s_x - \eta \alpha_x) \frac{2}{3} - (s_y - \eta \alpha_y) \frac{1}{3} - (s_z - \eta \alpha_z) \frac{1}{3} \right]
\]

Similarly, differentiation with respect to \( \sigma_y, \sigma_z \) and shear stresses can be accomplished.
From Eq. (1.9), it can be shown that $2S_x - S_y - S_z = 3S_x$ and similarly for other deviatoric normal stresses and corresponding $\alpha$ terms, the term needed (to calculate $[P]$) in Eq. (1.43) is obtained as

$$
\left\{ \frac{\partial F}{\partial \sigma} \right\} = \frac{3}{\eta \sigma_y + (1 - \eta)\sigma_0} \left[ \begin{array}{c} S_\sigma - \eta \alpha \varepsilon \\ 0 \\ S_z - \eta \alpha _z \\ \end{array} \right] + \left\{ \begin{array}{c} 0 \\ \end{array} \right\}
$$

(1.46)

where, $\{\alpha\} = \left\{ \begin{array}{c} \alpha_S \\ \alpha_z \\ \end{array} \right\}$

Using the Eqs. (1.37) (1.43) and (1.46), $[E_{ep}]$ and $[K_\alpha]$ can be calculated.

### 1.12. SCOPE OF THE PRESENT RESEARCH WORK

In this research work, only isotropic unstiffened thin plates are considered for analysis. Simply supported boundary conditions are assumed in all the analysis. The uniaxial compression loading is applied as uniform edge displacement. Shell181 (a 4-noded quadrilateral element of ANSYS) elements are used for analysis. The present investigation deals with the numerical determination of ultimate strength of thin square plates with geometrical imperfections, the scope of which is summarized below.

1. **Effect of distributed (random) geometrical imperfections on the ultimate strength of thin HT-32 steel square plates (of size 1000 x 1000 x 8 mm) and reliability calculations of imperfect plates using Mean Value First Order Second Moment (MVFOSM) method.**

2. **Effect of a centrally located dent in terms of its size (dent length, dent width and dent depth) and angle of orientation on the ultimate strength of thin HT-32 steel square plates (of size 1000 x 1000 x10 / 12 / 16 / 20 mm and 500 x 500 x 5 / 8 / 10 mm).**

3. **Interaction effect of two short dents of same size in terms of their location, orientation and distance between their centres (centre
distance) on the ultimate strength of thin HT-32 steel square plates (of size 1000 x 1000 x 12 mm and 500 x 500 x 8 mm),

4 Effect of a centrally located dent in terms of its size (dent length, dent width and dent depth) and angle of orientation on the ultimate strength of thin Al 2024-T3 alloy square plates (of size 500 x 500 x 5 / 8 / 10 mm), and

5 Interaction effect of two short dents of same size in terms of their location and centre distance on the ultimate strength of thin Al 2024-T3 alloy square plates (of size 500 x 500 x 8 mm).

1.13. OBJECTIVES OF THE PRESENT RESEARCH WORK

The main objective of this thesis is to evaluate the effect of distributed random geometrical imperfections and localized geometrical imperfections - dents on the static ultimate collapse strength (hereafter referred as ultimate strength) of the thin plates under uniaxial compression.

The specific objectives of this thesis are:

- To develop a simple methodology for designers to determine the reliable safe load on the thin plates under uniaxial compression based on distribution of ultimate strengths due to distributed random geometrical imperfections.

- To determine the effect of variation of size and orientation of localized geometrical imperfections viz. dents on ultimate strength of thin HT-32 steel and Al 2024-T3 alloy plates under uniaxial compression.

- To study the interaction effect of two short dents of same size, by varying the location, orientation and centre distance between the dents, on ultimate strength of thin HT-32 steel and Al 2024-T3 alloy plates under uniaxial compression.
1.14. ORGANIZATION OF THESIS

Chapter 1 discusses about types of buckling, bifurcation behaviour of thin plates, and effect of geometrical imperfections on ultimate strength, introduction about the FE non linear elasto-plastic analysis. Also the scope, main objectives and general methodology adopted in the investigation are presented.

Chapter 2 presents the review of literature related to subject of study.

Chapter 3 discusses about the analytical solution used in the study.

Chapter 4: In this chapter, method adopted to generate random distributed geometrical imperfections by using linear combination of eigen affine mode shapes of perfect plate and results obtained from non linear FE analysis of imperfect plates are presented. Further, a simple reliability calculation procedure proposed based on mean value first order second moment method (MVFOSM) to predict safe reliable load of thin plates is elaborated.

Chapter 5: In this chapter, modeling of single dent is presented. The effects of variation of size and orientation of a centrally located dent on ultimate strength of thin HT-32 steel square plates are also discussed with FE analysis results.

Chapter 6: In this chapter, modeling of two dents with varying centre distance is presented. The interaction effect of two short dents of same size (by varying their location, orientation and centre distance) on ultimate strength of thin HT-32 steel square plates are also discussed with FE analysis results.

Chapter 7: The effects of variation of size and orientation of a centrally located dent on ultimate strength of thin Al 2024-T3 alloy square plates are discussed with FE analysis results.

Chapter 8: The interaction effect of two nearly spherical short dents of same size (by varying their location, orientation and centre distance) on ultimate strength of thin Al 2024-T3 alloy square plates are discussed with FE analysis results.

Finally, conclusions and directions for further research are presented in Chapter 9.