CHAPTER 3

MODELLING

3.1 Introduction

In the domain of numerical methods, finite element method is an efficient method to analyze a complex structure. Discretization of the structure into small elements enables one to model structures with various complexities. Due to this, finite element softwares in the recent years have become more efficient in performing displacement and stress analysis in the linear and non-linear field, detailed study of damage, geometric non-linear and material non-linear of structures as well. Although the linear analysis requires lesser computational effort to produce results, non-linear analyses are required to capture realistic behaviour of structures. In the current study, non-linear explicit dynamic analyses are carried out using Abaqus/Explicit as the main tool for the study of dynamic buckling behaviour and the failure of laminated composite plates and cylindrical panels. Abaqus/Standard is used to calculate the static buckling analysis, frequency analysis and the non-linear static analysis (Riks analysis). In this chapter, first, the details of the finite element analysis used in the study are discussed and then, the mathematical formulations are presented. The problem considered in the present investigation is defined. The geometry, pulse loading function, material properties and the boundary conditions of the laminated composite plate and cylindrical panel considered are described. The dynamic buckling criterion considered in the investigation is also presented. The chapter is divided into the following subsections:

- Finite Element Modelling
    - Shell Element
    - Modelling Composite Layup
- Governing Equations
    - Static Buckling
    - Vibration

  - Post-Buckling
  - Dynamic Buckling
  - Failure Studies
- Problem description
  - Laminated Composite Plate
  - Laminated Composite Cylindrical Panel
  - Laminated Composite Cylindrical Panel with Cutout
  - Laminated Composite Stiffened Cylindrical Panel
- Dynamic Buckling Criterion

3.2 Finite Element Modelling

The details of the finite element modelling and the procedure for the analysis are described in this section. The shell element and the non-linear procedure used in the current investigation are described.

3.2.1 Shell element

Based on the geometry of the structure, discretization can be done in many ways. For this, the shape, size and arrangement are decided to simulate the model in the best possible manner. The shells elements can have 3, 4, 6,8 and 9 nodes. Each shell element has at least one integration point at which the stresses are calculated. The displacements are calculated at the nodes. In the current investigation, the shell elements used are S4R and S3 elements. S4R is a four-node quadrilateral element having reduced integration with hourglass control and finite membrane strains. In Abaqus, S4R element is used to model thick shells using first-order shear deformation theory, which is useful for large deformation analysis (Tarfaoui *et al.*, 2008). This element follows a reduced integration rule. Hourglass stabilization is achieved by using an hourglass stabilization parameter, which is necessary when a reduced integration scheme is involved (Laulusa *et al.*, 2006). S3 element is a three noded triangular element with finite membrane strains. At each node of the S4R and S3 elements, there are 6 degrees of freedom. The S4R element and the S3 element are shown in Fig. 3.1(a) and Fig. 3.1(b) respectively.

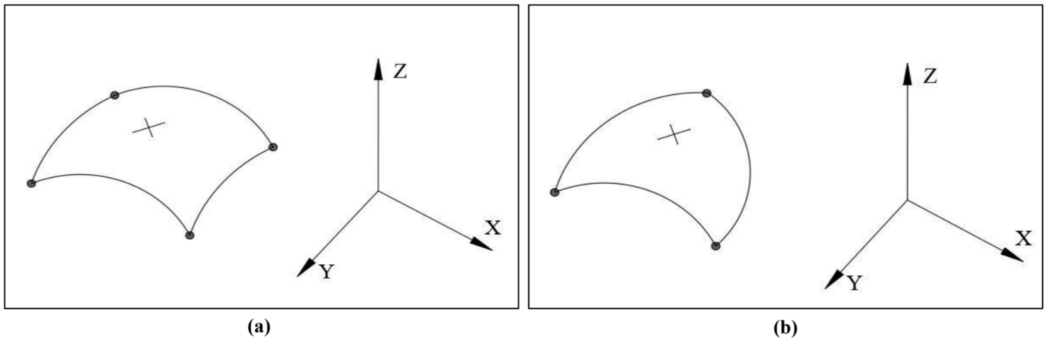

Fig. 3.1 Shell elements **(a)** S4R element **(b)** S3 element

In the case of cylindrical panels without stiffeners and cutouts, only S4R elements are used. The whole domain is divided into the same number of divisions for which the convergence and validation studies are performed. In the case of panels with cutouts, both S4R and S3 elements are used to model the cylindrical panel. Furthermore, in the case of stiffened cylindrical panels, both S4R and S3 elements are used. The mesh density at the junction between the skin and the stiffener is higher than at the edges, so that, the failure at the junction could be recognized.

3.2.2 Modelling composite layup

The laminated composite panels can be modelled in Abaqus/Standard and Abaqus/Explicit. Figure 3.2 (a) shows a typical laminated composite plate. In this figure, ‘1’, ‘2’ and ‘3’ are the principal material direction. Furthermore, ‘1’ direction aligns with the global Y-axis which is the loading direction in the present investigation. Also, ‘3’ represents the normal direction and the piles are oriented in counter-clockwise about ‘3’. This means, 0° plies align with ‘1’ direction and 90° plies align with ‘2’ direction. The stacking begins from the bottommost ply in the increasing thickness direction. In each ply, three integration points are considered. In the failure analysis, the failure index can be maximum at any of the plies and the maximum failure index is considered in the first ply failure analysis of the plate and cylindrical panel.

For modelling the stiffened cylindrical panel, the geometry of the panel and the skin are drawn as shells, and the geometry is extruded. For stacking scheme in the skin and the stiffener, either parallel stacking scheme (Fig. 3.2(b)) or perpendicular stacking scheme (Fig. 3.2(c)) can be considered.

The geometric non-linearity is incorporated in the analysis by considering the non-linear kinematic relationship occurring due to large displacements in the structure. The analysis results depend on the previous steps, which means that the calculations are performed at local coordinates and are updated and used in the subsequent steps (Abaqus, 2013). The shell element described in this section is used to solve the governing equations described in the next section.

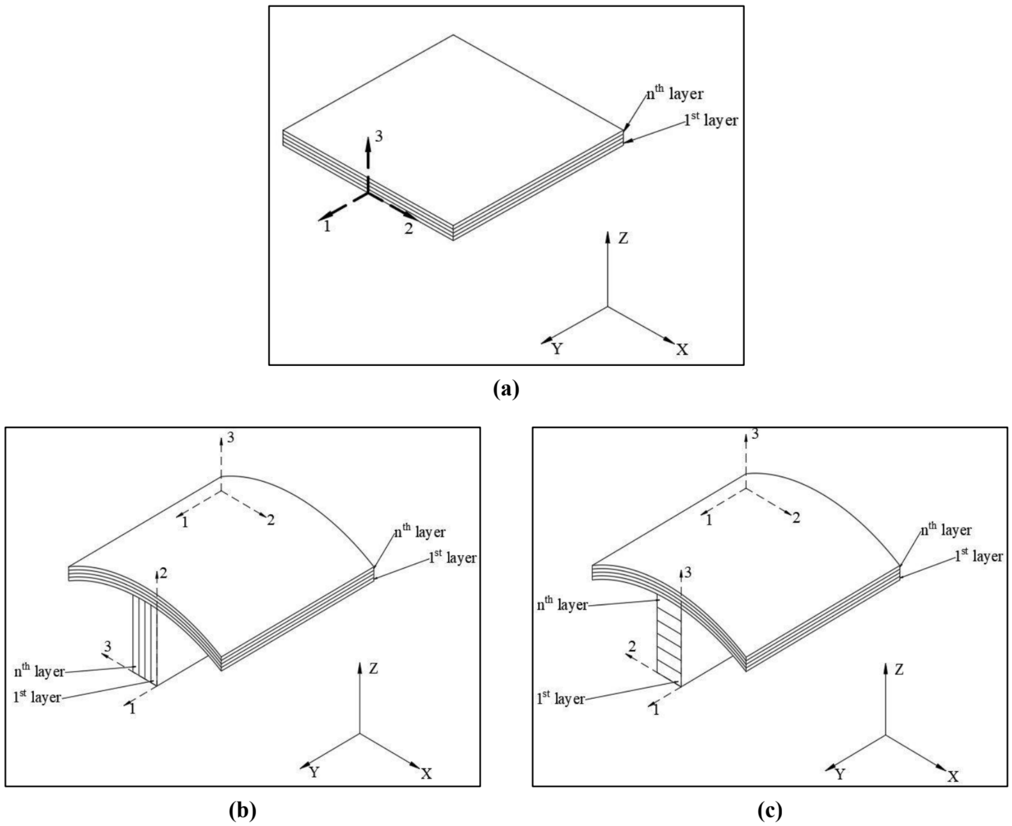

Fig. 3.2 Stacking scheme in **(a)** Laminated composite plate **(b)** Parallel stacking scheme in a stiffened cylindrical panel **(c)** Perpendicular stacking scheme in a stiffened cylindrical panel

3.3 Governing Equations

The governing equations solved using Abaqus software for static buckling, vibration, post-buckling, dynamic buckling and failure studied are presented in this section. The governing equation of motion is given in Eqn. (3.1). The damping is not considered in the study. This is a general equation of motion which can be reduced to get the equations for static buckling, vibration and dynamic buckling analysis.

$$[M]\{\ddot{u}\} + [K(\{u\})]\{u\} = \{F(t)\} \tag{3.1}$$

Where,

$$[M] = \int N^T \rho\, N\, dx\, dy\, dz$$

$$[K] = \int ([B]_L + [B]_{NL})^T [D] \left([B]_L + \frac{1}{2}[B]_{NL}\right) dxdydz$$

$$\{F\} = \int N^T b_0\, dV + \int N^T \tau\, dS + \int N^T \bar{p}\, dA$$

In Eqn. (3.1), $[M]$ is the mass matrix,

$[K]$ is the stiffness matrix,

$\{F(t)\}$ is the load vector.

$\{\ddot{u}\}$ is the nodal acceleration vector,

$\{u\}$ is the nodal displacement vector,

ρ is the mass density,

τ *is the* boundary traction over the surface,

b_0 is the body force per unit volume,

$\bar{p}$ is the pressure acting on the surface,

N is the shape function matrix and

B_L is the linear strain-displacement matrix

B_{NL} is the non-linear strain-displacement matrix

The stiffness matrix is a function of deformation since geometric non-linearity is considered in the current study.

3.3.1 Static buckling

The governing equation for static buckling problem is given in Eqn. (3.2), where $[K_G]$ is the geometric stiffness matrix and P_{cr} is the critical buckling pressure.

$$\left[[K] - P_{cr}[K_G]\right]\{u\} = 0 \tag{3.2}$$

The solution of Eqn. (3.2) in Abaqus/Standard provides the eigenvalue which multiplied by the load factor gives the critical buckling load of the plate or the cylindrical panel. Before the commencement of buckling analysis, the structure can have some preloads in the form of dead loads or loads due to temperature change. This is considered as the base state of the structure relative to which the buckling loads are calculated. Usually, if the structure is at rest, these preloads are zero. The critical buckling pressure is the total of preloads and the eigenvalue in the elastic range of the structure. For extracting the eigenvalue, subspace iteration technique is used which is efficient for extracting eigenmodes lesser than 20.

3.3.2 Vibration

The governing equation for calculating the natural frequency is given in Eqn. (3.3), where, the ω_n is the natural frequency.

$$\left[[K] - \omega_n^2[M]\right]\{u\} = 0 \tag{3.3}$$

The solution of Eqn. (3.3) in Abaqus/Standard is a linear perturbation procedure to extract the natural frequency and mode shapes of the structure. In the current study, the first natural frequency is used to calculate the first natural period of plate and cylindrical panel, which is utilized to fix the duration of loading in the dynamic buckling analysis.

3.3.3 Post-buckling

The non-linear static displacement is evaluated using post-buckling analysis. The governing equation is given in Eqn. (3.4).

$$[K\{u\}]\{u\} = \{F\} \tag{3.4}$$

The above equation (Eqn. 3.4) is solved in Abaqus/Standard using Riks analysis. Riks analysis is used to investigate the geometric non-linear behaviour of the structure.

Abaqus/Standard uses the arc-length method along the equilibrium path to solve for loads and displacements in Eqn. (3.4) simultaneously.

3.3.4 Dynamic buckling

The equation used to solve the dynamic buckling problem is given in Eqn. (3.5). The damping in the plate and cylindrical panel is not considered.

$$[M]\{\ddot{u}\} + [K(\{u\})]\{u\} = \{F(t)\} \tag{3.5}$$

The above equation (Eqn. 3.5) is solved using Abaqus/Explicit. In explicit dynamic analysis, the displacements and velocities are calculated in terms of quantities that are known at the beginning of an increment; therefore, the global mass and stiffness matrices need not be formed and inverted, which means that each increment is relatively inexpensive compared to the increments in an implicit integration scheme. The time integration is done using the central difference method where Eqn. (3.5) is solved at the beginning of the increment *t* the accelerations calculated at time *t* are used to advance the velocity solution to time ($t + \Delta t/2$) and the displacement solution to time ($t + \Delta t$). The central-difference integration operator is explicit in the sense that the kinematic state is advanced using known values of velocity and acceleration from the previous increment.

3.3.5 Failure studies

The first ply failure of the composite plate and cylindrical panel is calculated and compared with the dynamic buckling load. The failure criteria used in the current study are Tsai -Wu criterion, Tsai Hill criterion, Azzi-Tsai-Hill criterion and Maximum stress criterion. The equations used for failure theories are discussed in the subsequent section. In these equations (Eqn.3.6-3.9), σ_{11} and σ_{22} are the normal stresses in the principal material direction and transverse-to material direction. Also, σ_{12} is the shear stress.

3.3.5(a) Azzi-Tsai-Hill theory

The Azzi-Tsai-Hill failure is given by Eqn. (3.6).

$$Failure\ Index = \frac{\sigma_{11}^2}{X^2} - \frac{|\sigma_{11}\sigma_{22}|}{X^2} + \frac{\sigma_{22}^2}{Y^2} + \frac{\sigma_{12}^2}{S^2} < 1 \qquad (3.6)$$

3.3.5(b) Maximum Stress theory

If $\sigma_{11}>0$, $X=X_T$; otherwise, $X=X_C$. If $\sigma_{22}>0$, $Y=Y_T$; otherwise, $Y=Y_C$. The maximum stress failure is given by Eqn. (3.7).

$$Failure\ Index = max\left(\frac{\sigma_{11}}{X}, \frac{\sigma_{22}}{Y}, \left|\frac{\sigma_{12}}{S}\right|\right) < 1 \qquad (3.7)$$

3.3.5(c) Tsai-Hill theory

If $\sigma_{11}>0$, $X=X_T$; otherwise, $X=X_C$. If $\sigma_{22}>0$, $Y=Y_T$; otherwise, $Y=Y_C$. The Tsai-Hill Criterion is given by Eqn. (3.8).

$$Failure\ Index = \frac{\sigma_{11}^2}{X^2} - \frac{\sigma_{11}\sigma_{22}}{X^2} + \frac{\sigma_{22}^2}{Y^2} + \frac{\sigma_{12}^2}{S^2} < 1 \qquad (3.8)$$

3.3.5(d) Tsai-Wu theory

The Tsai-Wu failure criterion is given by Eqn. (3.9).

$$Failure\ Index = F_1\sigma_{11} + F_2\sigma_{22} + F_{11}\sigma_{11}^2 + F_{22}\sigma_{22}^2 + F_{66}\sigma_{12}^2 + 2F_{12}\sigma_{11}\sigma_{22} < 1 \qquad (3.9)$$

The coefficients are defined as:

$$F_1 = \frac{1}{X_T} + \frac{1}{X_T} \qquad F_2 = \frac{1}{Y_T} + \frac{1}{Y_C} \qquad F_{11} = \frac{-1}{X_T X_C}$$

$$F_{22} = \frac{-1}{Y_T Y_C} \qquad F_{66} = \frac{1}{S^2} \qquad F_{12} = \bar{f}\sqrt{F_{11}F_{22}}$$

In Eqn. (3.9), the value of $\bar{f} = -0.5$ which is for the material considered in the current study as suggested in Hinton *et al.* (2004).

3.4 Problem description

In the present investigation, the stability and the first ply failure of five cases are considered:

- Laminated composite plate
- Laminated composite cylindrical panel
- Laminated composite cylindrical panel with cutout
- Laminated composite stiffened cylindrical panel

In the succeeding sections, the geometry, pulse loading function, material properties and the boundary conditions of the laminated composite plate and cylindrical panel considered are described. The dynamic buckling criterion considered in the investigation is also presented.

3.4.1 Laminated composite plate

The geometry, material properties, pulse loading type and the boundary conditions of the laminated composite plate are described in this section. The geometry of the plate is shown in Fig. 3.3(a) where '*a*' is the length of the loaded edge, '*b*' is the length of the unloaded edge and '*h*' is the thickness of the plate. Simply supported boundary conditions are considered as shown in Fig. 3.3(b). In Fig. 3.3(b), '*u*', '*v*' and '*w*' are the displacements along X-axis, Y-axis and Z-axis respectively; 'θ_X' corresponds to the rotation about X-axis, 'θ_Y' corresponds to the rotation about Y-axis and 'θ_Z' corresponds to the rotation about Z-axis. The material properties considered are from Narita and Leissa (1990) and Hinton *et al.* (2004) shown in Table 3.1 and Table 3.2 respectively.

In Table 3.1 and Table 3.2, E_{11} is the principal Young's modulus in the material direction, E_{22} is the principal Young's modulus in the transverse-to material direction, G_{12} is the shear modulus associated with plane 1-2, G_{23} is the shear modulus associated with plane 2-3 and G_{13} is the shear modulus associated with plane 1-3. Furthermore, ν_{12} is the major Poisson's ratio and ν_{21} is the minor Poisson's ratio. Apart from these, X_T is the tensile strength of the lamina in the principal material direction, X_C is the compressive strength of the lamina in the principal material direction, Y_T is the tensile strength of the lamina in the transverse-to principal material direction and Y_C is the compressive strength of the lamina in the transverse-to principal material direction. S_{12} is the shear strength of the lamina in plane 1-2, S_{23} is the shear strength of the lamina in plane 2-3 and S_{13} is the shear strength of the lamina in plane 1-3.

Perfect plates are very difficult to manufacture. Some amount of imperfection will always be present in a plate and should be considered in the analysis. Measurement of imperfections is a difficult process. In the current study, the value of the imperfection considered is a certain percentage of the thickness of the plate. The shape of imperfection is the first buckling mode the plate as shown in Fig. 3.4. With the change in stacking sequence or the geometry, the first buckling mode of the plate changes. However, for uniformity, the same shape of imperfection is considered for all cases of stacking sequences (Fig. 3.4).

Table 3.1 Material properties of Graphite/Epoxy lamina (Narita and Leissa, 1990)

Property	**Value**
E_{11}	138 GPa
E_{22}	8.96 GPa
G_{12}	7.1 GPa
G_{23}	7.1 GPa
G_{13}	7.1 GPa
ν_{12}	0.3
ν_{21}	0.0195

Table 3.2 Material properties of T300/BSL914C lamina (Hinton *et al*., 2004)

Property	**Value**
E_{11}	138 GPa
E_{22}	11 GPa
G_{12}	5.5 GPa
G_{23}	5.5 GPa
G_{13}	5.5 GPa
υ_{12}	0.28
X_T	1500 MPa
X_C	900 MPa
Y_T	27 MPa
Y_C	200 MPa
S_{12}	80 MPa
S_{23}	80 MPa
S_{13}	80 MPa

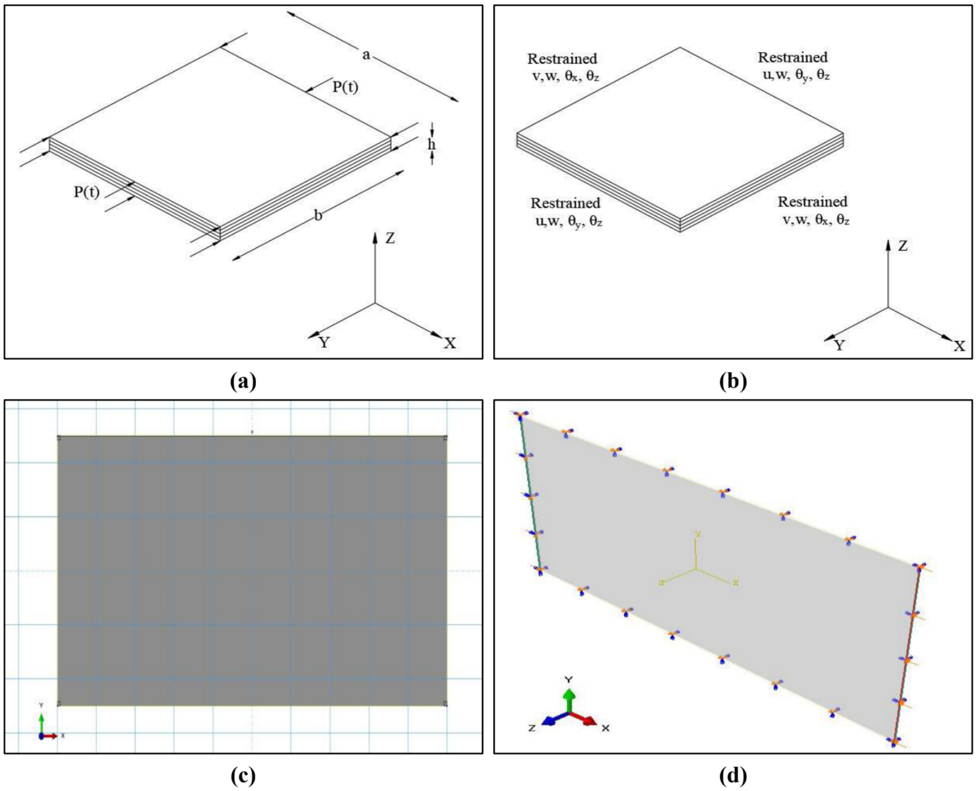

Fig. 3.3 Laminated composite plate **(a)** Geometry **(b)** Simply supported boundary conditions **(c)** Geometry of rectangular plate in Abaqus **(d)** Model analyzed in Abaqus

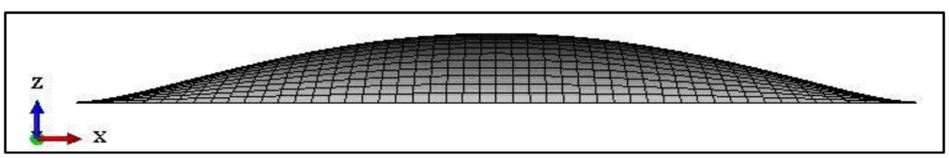

Fig. 3.4 Plate with imperfection in the form of first mode shape

The stacking sequence is (θ°/-θ°/ θ°) (Narita and Leissa, 1999). Another set of stacking sequence considered is balanced symmetric cross-ply laminates (0°/90°/90°/0°) and balanced symmetric angle ply laminates (45°/-45°/-45°/45°) from Hinton *et al.* (2004). The stacking sequence from Narita and Leissa (1999) is used for dynamic buckling studies only. The second set of stacking sequence from Hinton *et al.* (2004) is used to carry out both the dynamic buckling analysis and the failure analysis. Three types of loading functions are considered: Rectangular pulse loading (Fig. 3.5a), Sinusoidal pulse loading (Fig. 3.5b) and Triangular pulse loading (Fig. 3.5c). The magnitude of the dynamic load is the maximum amplitude of the dynamic load for the corresponding pulse loading function considered as shown in Fig. 3.5(a)-Fig.3.5(c).

In order to model the laminated composite plate, the geometry of the plate is drawn after which the material properties and the required property assignments are done. Depending upon the type of analysis, either buckling analysis, frequency analysis or dynamic analysis are defined. The loading conditions (amplitude and duration for dynamic analysis) and the boundary conditions are defined. The analysis is carried out and the post-processing of the results is done. The geometry of the plate drawn in Abaqus is shown in Fig. 3.3(c) and the model with applied loads and the boundary conditions are shown in Fig. 3.3(d).

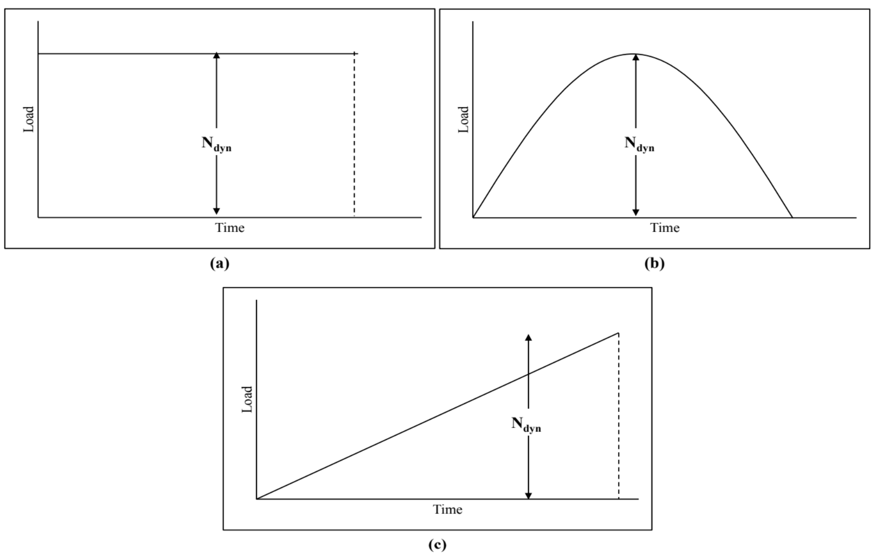

Fig. 3.5 Pulse loading functions **(a)** Rectangular **(b)** Sinusoidal **(c)** Triangular

3.4.2 Laminated composite cylindrical panel

The geometry, material properties, pulse loading type and the boundary conditions of the composite cylindrical panel are described in this section. The geometry of the panel is shown in Fig. 3.6. The material properties considered are from Hinton et al. (2004) shown in Table 3.2. Four types of boundary conditions are considered: Simply supported on all four sides (Fig. 3.7(a)), two clamped edges on the non-loaded edges and simply supported on the other edges

(Fig. 3.7(b)), one non-loaded edge clamped, and three edges simply-supported (Fig. 3.7(c)) and one non-loaded edge free and other edges simply-supported (Fig. 3.7(d)). The panel is subjected to rectangular and sinusoidal pulse loads (Fig. 3.5(a) and Fig. 3.5(b)).

In Abaqus, the geometry of the cylindrical panel is drawn with the help of the circle and trimmed to the appropriate chord length. The central rise of the panel is calculated based on the length of the side *a* (Fig. 3.6(a)). The geometry of the arc is then extruded to the required length so as to get the panel with required dimensions. After the geometry of the panel is drawn, the material properties and the required property assignments are done. Depending upon the type of analysis, either buckling analysis, frequency analysis or dynamic analysis are defined. The loading conditions (amplitude and duration for dynamic analysis) and the boundary conditions are defined. The analysis is carried out and the post-processing of the results is done. The geometry of the cylindrical panel drawn in Abaqus is shown in Fig. 3.6(b) and the model with applied loads and the boundary conditions are shown in Fig. 3.6(c).

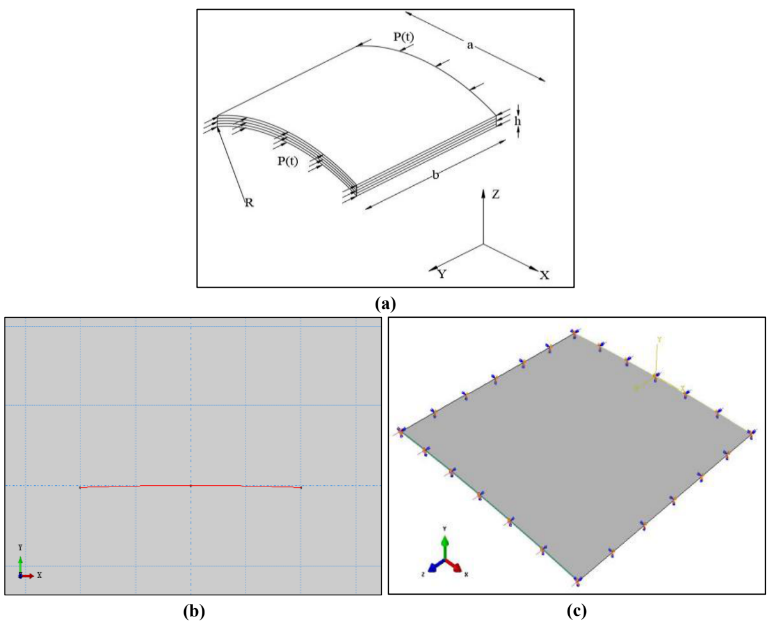

Fig. 3.6 Laminated composite cylindrical panel **(a)** Geometry **(b)** Geometry of the curve extruded in Abaqus **(c)** Model analyzed in Abaqus

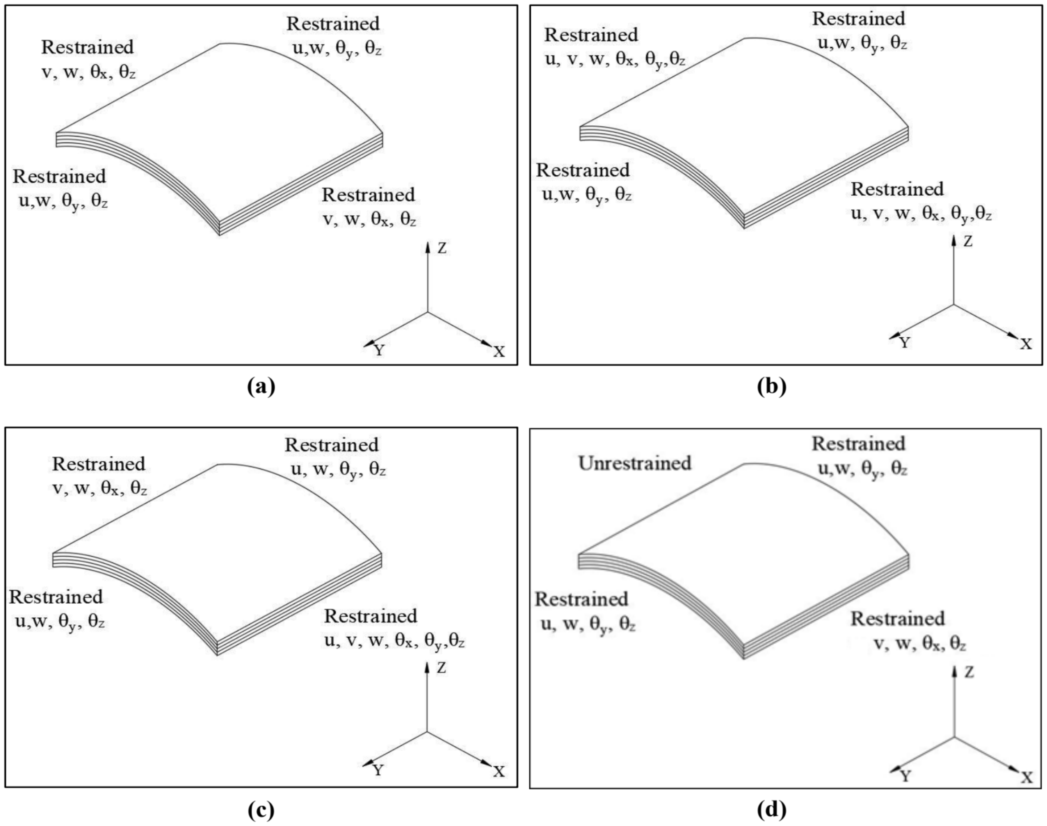

Fig. 3.7 Boundary conditions for the cylindrical panel **(a)** Simply supported boundary conditions (BC1). **(b)** Two edges simply supported and two clamped (BC2). **(c)** Three edges simply supported, and one edge clamped (BC3). **(d)** Three edges simply supported, and one edge free (BC4).

3.4.3 Laminated composite cylindrical panel with cutout

The geometry, material properties, pulse loading type, cutout geometry and the boundary conditions of the composite cylindrical panel with cutouts are described in this section. The geometry of the panel with a general cutout is shown in Fig. 3.8(a). The material properties considered are from Hinton *et al.* (2004) shown in Table 3.2. Simply supported boundary conditions are considered (Fig. 3.8(b)). The panel is subjected to rectangular and sinusoidal pulse loads. Three shapes of the cutout are considered. The plans of circular, square and square-rotated shape cutouts are shown in Fig. 3.9(a)-3.9(c).

In Abaqus, the geometry of the cylindrical panel is drawn first and extruded to the required length. Then a plane at which the cutout is required is placed and the required geometry of the cutout is drawn. In the present investigation, the cutout is provided at the center of the panel, so the geometry of the pane is drawn at the plane passing through the entire thickness of the panel. After the geometry of the panel is drawn, the material properties and the required property assignments are done. Depending upon the type of analysis, either buckling analysis, frequency analysis or dynamic analysis are defined. The loading conditions (amplitude and duration for dynamic analysis) and the boundary conditions are defined. The analysis is carried out and the post-processing of the results is done. The geometry of the cylindrical panel with circular cutout drawn in Abaqus is shown in Fig. 3.8(c) and the model with applied loads and the boundary conditions are shown in Fig. 3.8(d).

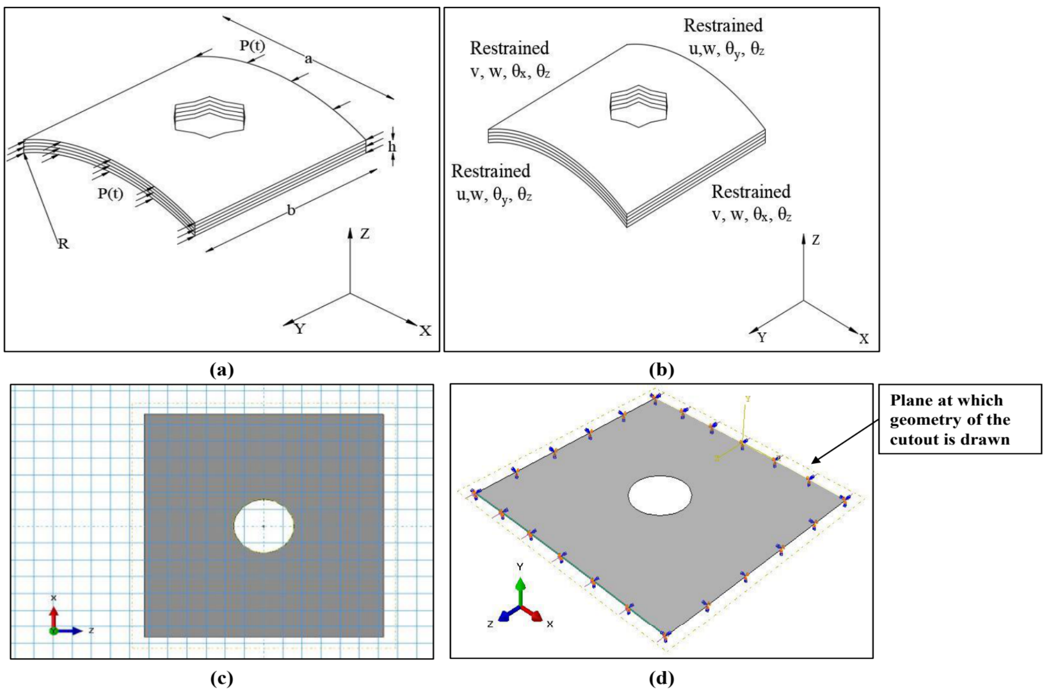

Fig. 3.8 Laminated composite cylindrical panel with a general shaped cutout **(a)** Geometry **(b)** Simply supported boundary conditions **(c)** Geometry of the panel with cutout drawn in Abaqus **(d)** model analyzed in Abaqus

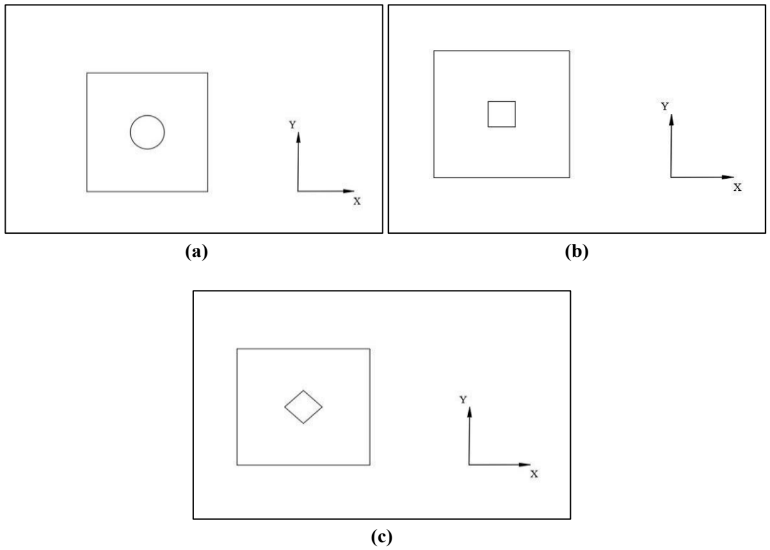

Fig. 3.9 Plan of the cutout shape **(a)** Circular **(b)** Square **(c)** Square-Rotated

3.4.4 Laminated composite stiffened cylindrical panel

The geometry, material properties, pulse loading type and the boundary conditions of the stiffened cylindrical panel are described in this section. The geometry of the stiffened panel is shown in Fig. 3.10(a). The stacking sequence in the skin and the stiffeners are perpendicular to each other as shown in Fig. 3.2(b). In Fig. 3.2(b), 'n' represents the normal direction and the piles are oriented in counter-clockwise about 'n'. This means, 0° aligns with '1' direction and 90° aligns with '2' direction. The material properties considered are from Hinton et al. (2004) shown in Table 3.2. Simply supported boundary conditions are considered (Fig. 3.10(b)). The boundary conditions are applied for both the skin edges and the stiffener ends parallel to Y-axis. The panel is subjected to rectangular and sinusoidal pulse loads (Fig. 3.5(a) and Fig. 3.5(b)).

In Abaqus, the geometry of the cylindrical panel along with the stiffener is drawn with both the skin and the stiffener in contact with each other and extruded to the required length. After the geometry of the panel is drawn, the material properties and the required property assignments are done. Depending upon the type of analysis, either buckling analysis, frequency analysis or dynamic analysis are defined. The loading conditions (amplitude and duration for dynamic analysis) and the boundary conditions are defined. The analysis is carried out and the post-processing of the results is done. The geometry of the stiffened cylindrical panel drawn in Abaqus is shown in Fig. 3.10(c) and the model with applied loads and the boundary conditions are shown in Fig. 3.10(d).

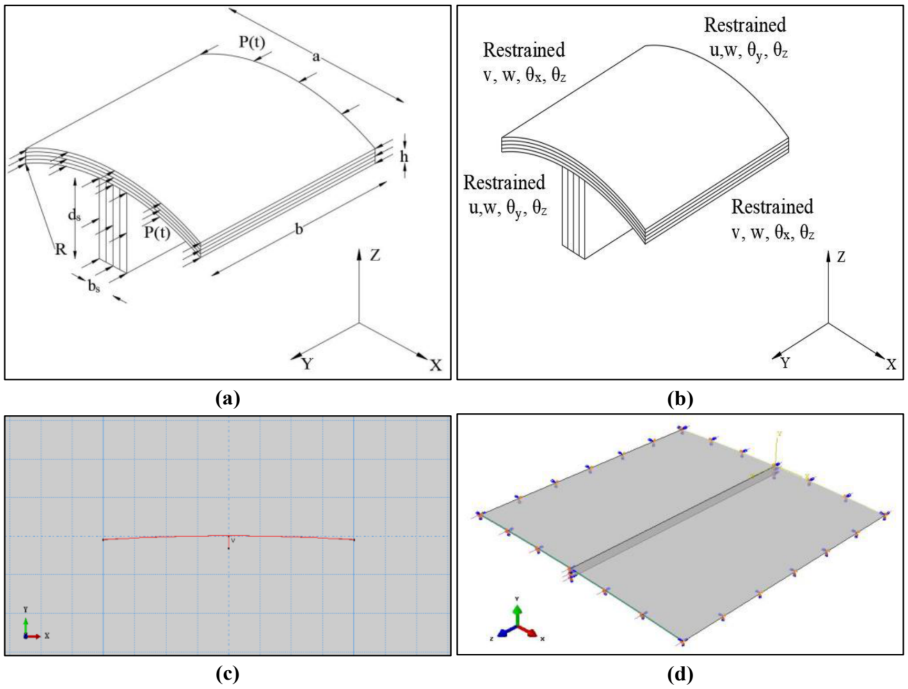

Fig. 3.10 Laminated composite stiffened cylindrical panel **(a)** Geometry **(b)** Simply supported boundary conditions **(c)** Geometry of the panel with stiffener drawn in Abaqus **(d)** model analyzed in Abaqus

3.5 Dynamic Buckling criterion

The dynamic buckling load is calculated using Vol'mir criterion where the dynamic buckling load is the in-plane dynamic load at which the transverse displacement is equal to the thickness of the panel (Vol'mir, 1974). The results obtained from this criterion are within 10% variation with the results obtained from Budiansky-Hutchinson criterion (Kubiak, 2013; Kowal-Michalska, 2010). Also, in the case of stiffened cylindrical panels, it is not possible to apply Budiansky-Hutchinson criterion as rapid growth in displacements is not observed (Patel *et al.*, 2011). Furthermore, for large imperfections in the stiffened plate, Budiansky-Roth criterion is not suitable as the dynamic buckling criterion (Tao *et al.*, 2004b). Hence, Vol'mir's criterion is considered in the present investigation. In the current study, the ratio w/h=1 is critical and the magnitude of load at which this ratio is reached is the non-linear dynamic buckling load.

3.6 Summary

The finite element model and the governing equations described in this chapter are used to analyze the static buckling, vibration, post-buckling, dynamic buckling behaviour and failure of the laminated composite plates and cylindrical panels. The details of the present investigation including the geometry, pulse loading conditions, material properties and the boundary conditions are described in this chapter. In Chapters 4-7, the results of the convergence study, validation study and the results of the present study are presented. The conclusions drawn from each chapter is described in Chapter 8.