Composite Structures 93 (2011) 1363–1375

Contents lists available at ScienceDirect

Composite Structures

journal homepage: www.elsevier.com/locate/compstruct

Finite element modeling of low-velocity impact on laminated composite plates and cylindrical shells

S.M.R. Khalili [a,b,c,*], M. Soroush [b], A. Davar [a], O. Rahmani [a]

[a] Centre of Excellence for Research in Advanced Materials and Structures, Faculty of Mechanical Engineering, K.N. Toosi University of Technology, Tehran, Iran
[b] Islamic Azad University, South Tehran Branch, Department of Mechanical Engineering, Tehran, Iran
[c] Faculty of Engineering, Kingston University, London, UK

ARTICLE INFO

Article history:
Available online 20 October 2010

Keywords:
Low-velocity impact
Polymer matrix composite
Finite element modeling
Element type
Solution method
Mesh pattern

ABSTRACT

In this study, composite laminates and shell structures subjected to low-velocity impact are investigated by numerical analysis using ABAQUS finite element code. In order to model the impact phenomena by commercial finite element codes, various procedures are available. Accurate modeling requires the appropriate selection of element type, solution method, impactor modeling method, meshing pattern and contact modeling. In this investigation, by considering several case studies with various conditions, validity of the existed modeling processes is examined. In each case, by comparing the results of various methods with the related available experimental test results in existing literature, the best procedure is proposed which can serve as benchmark method in low-velocity impact modeling of composite structures for future investigations.

1. Introduction

Advanced composite structures, owing to their inherently high specific mechanical properties, are widely used as primary structures in many applications such as aerospace, sport equipments, pressure vessels and automobile parts. In many situations, these composite structures are likely to encounter impact by foreign object projectiles. It is well known that composites are very susceptible to transverse impact. Despite of their many virtues, these structures show a highly complex impact behavior and are very sensitive to non-visual damages that strongly influence their residual load bearing capability.

The behavior of composites under impact has been of significant concern in many advanced engineering structures and components and many researchers have made their efforts to analyze the impact dynamics of composite structures. Kumar et al. [1] studied the effect of impactor and laminate parameters on the impact response and impact-induced damages in graphite/epoxy laminated cylindrical shells using 3D finite element formulation. Her and Liang [2] used the ANSYS/LS-DYNA finite element software to calculate the transient response of the impact on composite laminates, cylindrical and spherical shells. Kim et al. [3] developed a 3D finite element code to describe dynamic and impact behavior and predict the impact-induced damage of shell-shaped structures. Oguibe and Webb [4] proposed a numerical model based on finite element displacement method that includes the effects of transverse shear deformation and a failure algorithm which describes the energy dissipation during the damage process for the purpose of impact analysis. Vaziri et al. [5] used a super finite element method that exhibits coarse-mesh accuracy to predict the transient response of laminated composite plates and cylindrical shells subjected to non-penetrating impact by projectiles. Maiti and Sinha [6] used higher order shear deformation theory and first order shear deformation theory to develop a finite element method to investigate the impact behavior of doubly curved laminated composite shells. Mili and Necib [7] studied experimentally the behavior of E-glass/epoxy laminated composite plates under impact of aluminum projectile at low velocities. Zhao and Cho [8] investigated the impact-induced damage initiation and propagation in the laminated composite shell under low-velocity impact. They applied a three dimensional eight-node nonconforming element with Taylor's modification scheme to analysis the interlaminar stress distribution and damage propagation. Lakshminarayana et al. [9] presented a numerical simulation of static indentation and lateral impact tests of laminated composite plates, using general-purpose FEM code ABAQUS. Cairns and Lagace [10] studied the influence of some parameters such as impactor mass, preload and material properties on the impact behavior of laminated composite plates analytically. Sun and Chen [11] investigated the impact response behavior of initially stressed composite laminated plate using the finite element method. They applied the experimentally obtained

* Corresponding author at: Faculty of Mechanical Engineering, K.N. Toosi University of Technology, Pardis St., Molasadra Ave., Vanak Sq., Tehran, Iran. Tel.: +98 21 84063208; fax: +98 21 88674748.

E-mail address: smrkhalili2005@gmail.com (S.M.R. Khalili).

0263-8223/$ - see front matter © 2010 Elsevier Ltd. All rights reserved.
doi:10.1016/j.compstruct.2010.10.003

contact law into the finite element program and used Newmark time integration algorithm for solving the time dependent equation of the plate and the impactor. Ghosh and Sinha [12] developed a finite element analysis procedure to predict the initiation and propagation of damages as well as to analysis damaged laminated composite plates under forced vibration and impact loads. Kistler and Waas [13] studied both experimentally and numerically, the response of the low-energy impact of laminated plates with cylindrical curves. These carbon/epoxy panels with quasi-isotropic properties were embedded at the four edges and impacted by a steel sphere falling from various heights. The finite element analysis (FEA) is carried out using ABAQUS software. Tarfaoui et al. [14] presented a FEA of static and dynamic tests on thick filament wound glass/epoxy tubes. They used certain models for validating material characteristics to predict their elastic behavior for static and dynamic loadings and further, they developed an impact model including material property degradation used for damage prediction. Swanson et al. [15] conducted a study to determine the strain response of carbon/epoxy cylinders subjected to impact loading. Experiments were carried out on two sizes of cylinders, that is, a small cylinder with 0.559 m length, 96.5 mm inside diameter and 1.63 mm wall thickness and a large cylinder with 1.89 m length, 319 mm inside diameter and 5.08 mm wall thickness, using an air gun impact apparatus. The cylinders and impact projectiles were designed, so that the applicability of scaling rules could be considered. They also carried out the theoretical analysis for the impact loading by a closed-form dynamic solution.

The purpose of this study is to provide a numerical treatment tool based on ABAQUS commercial finite element code for the reliable design of such structures under transverse impact loadings. The main aim of this work is to provide a general solution for the modeling of dynamic and quasi-static simulation of impact on the composite plate and shell structures and cylindrical shells with various side-to-thickness ratios by ABAQUS. In order to do this, several experimental and numerical examples [10–15] are selected for the modeling by FEM approach. These examples are classified in four separate geometries including: composite plate (Sun and Chen [11]), curved composite laminate (Kistler and Waas [13]), composite thick cylinder (Tarfaoui et al. [14]) and composite thin cylinder (Swanson et al. [15]). Next, a brief description of the methods and the procedures used is given and the evaluation of the accuracy of the element type, mesh pattern, impactor modeling and solution method for the analysis of composite structures is examined. Since, the authors intention in the present work is focused on dynamic response of the structures, the damage induced in the target structure is not considered in this work and the material behavior is assumed to be linear elastic.

2. Impact modeling procedure

This section, briefly describes the impact modeling process of composite structures. In this study, for simulation of the impact phenomena, the ABAQUS version 6.8-1 is used. Numerical analysis is often performed by the finite element solver ABAQUS/Explicit, which uses a central difference rule to integrate the equations of motion explicitly through the time [16]. The plates and shells are meshed using SC8R or S4R elements which are, respectively, an eight-node continuum shell and four-node conventional shell elements with reduced integration. The two types of the elements used for meshing plate or shell structures are shown in Fig. 1.

2.1. Meshing process and type of elements

Conventional shell, continuum shell and solid elements are available options in the modeling of the composite structures.

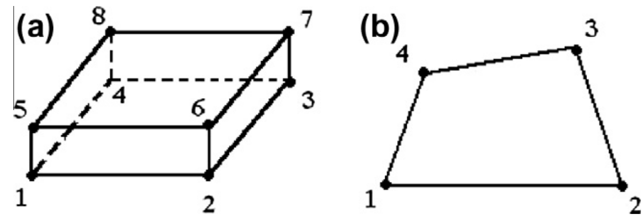

Fig. 1. (a) SC8R element and (b) S4R element [16].

S4R element, classified in the category of conventional shell elements, is a four-noded shell element which has six degrees of freedom per each node. It is based on the first order shear deformation theory and it has a good performance in large deformation analysis. It is included in the general-purpose elements category and suitable for thick and thin shell and plate structures [14,16].

SC8R element is classified in the category of continuum shell elements. This element is an eight-noded continuum shell element and has three degrees of freedom per each node (only displacement degrees of freedom). The choice of an element type used for the plates or shells as target structures depends on their side-to-thickness ratio as well as the impactor velocity. Increasing the thickness of target and impactor velocity leads to raising the shear deformation. Hence, in such cases, it is necessary to choose an appropriate element to avoid errors and to model the structure accurately [17–19].

According to side to the thickness ratio for each case study in these papers, both SC8R and S4R elements are examined in the examples of Sun and Chen [11] and Tarfaoui et al. [14]. For the other cases, i.e. Kistler and Waas [13], and Swanson et al. [15], due to their small values of the side-to-thickness ratios, only S4R elements are used. In Fig. 2, two elements types (S4R and SC8R) are shown which are used in modeling process of thick cylinders [14].

2.2. Impactor modeling

It is possible to model the impactor in three different methods: (1) impactor could be assumed as a rigid body and no property is assigned which means that the impactor is with infinite rigidity (which is referred as fully rigid in this paper), (2) impactor could be assumed as a rigid body, but with property assigned which means that the impactor is with real rigidity (∗RIGID BODY option), and (3) it is also possible to consider the impactor as a deformable body with property assigned. Using the last method, leads to a more realistic simulation of the impactor by using C3D8R element and the assigned properties like Young's modulus and density to the impactor as indicated in Fig. 3. Majority of researchers [13,14] assumed the impactor to be a rigid body. This assumption was considered, because of negligible deformation of the projectile.

2.3. Solution method

In order to model a dynamic phenomenon like impact by ABAQUS, it is possible to solve the problems with an explicit or implicit algorithm available in ABAQUS. Majority of researchers like Tarfaoui et al. [14] and Setoodeh et al. [20] solved the impact phenomena with implicit algorithm available in ABAQUS/Standard. But in this simulation, both the explicit and the implicit algorithms are used which are available in ABAQUS/Explicit and ABAQUS/Standard, respectively.

In an explicit scheme, the analysis cost rises only linearly with problem size, whereas the cost of solving the nonlinear equations associated with implicit integration rises more rapidly than linearly with problem size. Therefore, ABAQUS/Explicit is attractive

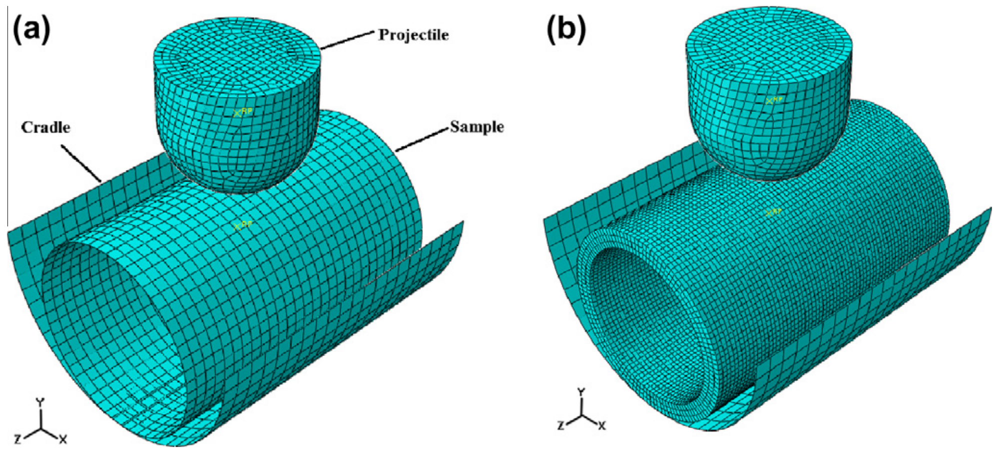

Fig. 2. FE modeling of composite cylinders by (a) S4R element and (b) SC8R element.

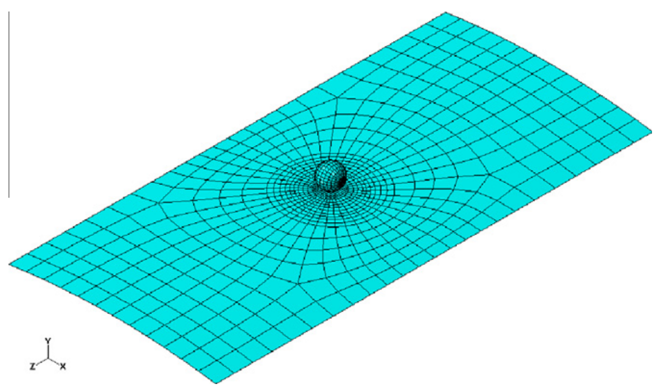

Fig. 3. Mesh used in the FE modeling of the impactor and the curved composite laminate.

for very large problems. This is shown in details during the modeling of selected case studies in the following sections.

In this study by comparing the various results obtained by explicit and implicit analysis, the appropriate procedure for modeling the low-velocity impact on composite structures is proposed.

2.4. Contact modeling

For contact modeling, there are many contact laws that can be applied in ABAQUS. In the present study, hard contact law is chosen. In this method, the contact constraint is applied when the clearance between two surfaces becomes zero. There is no limit in the contact formulation on the magnitude of contact pressure that can be transmitted between the surfaces [16]. The surfaces are separated when the contact pressure between them becomes zero or negative, then the constraint is removed. This behavior is referred to as ''hard'' contact [16].

The impactor and the target in the examples studied in this paper are set as the master surface and slave nodes, respectively. The contact force which is a function of the penetration distance is applied to the slave nodes to oppose the penetration, while equal and opposite forces act on the master surface at the penetration point [16].

2.5. Boundary and loading conditions

For definition of boundary conditions of the target structure, when it is required, the cylindrical coordinate system is defined first and then the appropriate boundary conditions are assigned to the middle surface of the shell. For example, this procedure for Swanson et al. [15] model is shown in Fig. 4. Also, in order to define the boundary conditions for the impactor, the movement of the impactor is restrained in all directions except translation along normal vector of the plate or the shell. The initial velocity of the impactor is specified as predefined field available in ABAQUS FE code at the reference point of the rigid impactor, or the whole impactor in the case of deformable impactors. Furthermore, in the case of rigid impactor, since no material properties are assigned, the mass of the impactor should be assigned at the reference point of the impactor.

2.6. Mesh pattern and convergence study

As it is clear, acquiring accurate results required fine mesh at vicinity of impact area. Some of the researchers like Ref. [3] used the structured mesh patterns as shown in Fig. 5a, but in this investigation, unstructured mesh pattern as shown in Fig. 5b is used. Comparison between the results of these two types of mesh patterns reveals that using unstructured mesh pattern like Fig. 5b, yields the results with almost the same order of accuracy, but with shorter elapsed CPU run-time, due to the less number of elements used in this pattern.

In order to confirm the applicability of the mesh pattern shown in Fig. 5b, in the example of Sun and Chen [11], different numbers of elements are examined as illustrated in Fig. 6. It is observed that by using coarser structured mesh pattern, many fluctuations are occurred in the history of the contact force. By increasing the number of elements (fine mesh), the fluctuations are decreased. The most accurate results are obtained for the optimum unstructured mesh pattern with 769 elements, as shown in Fig. 5b, taking CPU run-time equal to 38 s as well as the structured mesh pattern with 2304 (48×48) elements, as shown in Fig. 5a, taking CPU run-time equal to 62 s.

In order to obtain the optimum number of elements in unstructured mesh pattern, an analysis of the sensitivity of number of elements is performed. In Fig. 7a, mesh pattern of the circular area surrounding the contact region is shown. In Fig. 7b, the convergence study of the number of elements is performed. The variation of the maximum contact force versus the number of elements along the radius of the circular area surrounding the contact region in the unstructured mesh pattern is shown in this figure. In each case, the convergence study was performed and for brevity purpose, only the converged results are presented here.

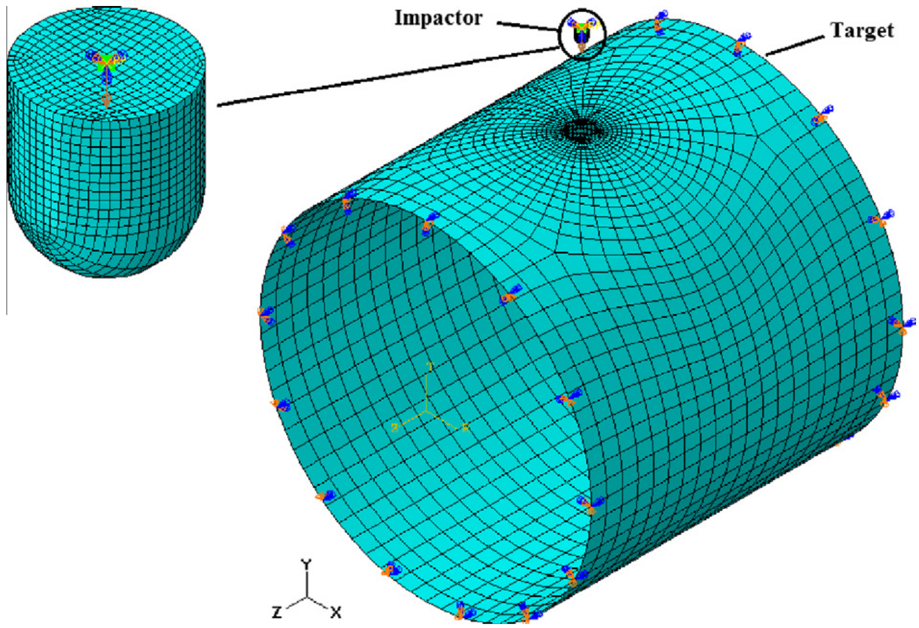

Fig. 4. Meshed impactor and thin cylinder with corresponding boundary conditions.

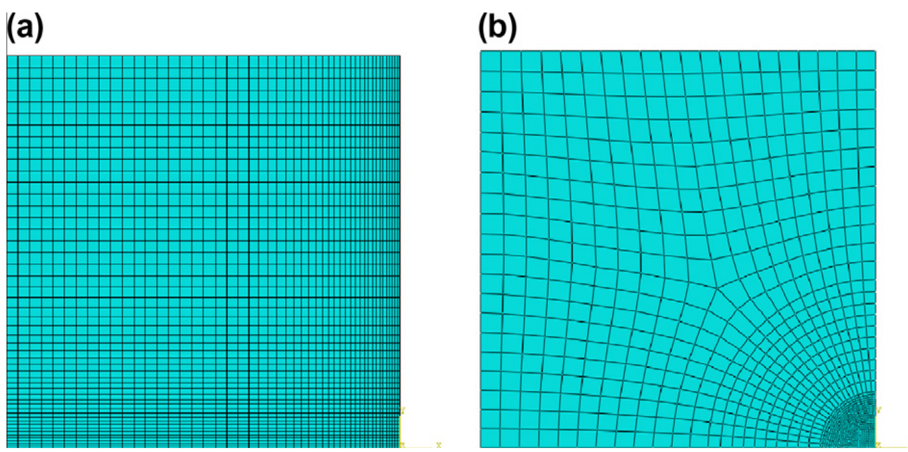

Fig. 5. Two types of mesh pattern for one quarter meshed plate model: (a) 48 × 48 structured mesh pattern and (b) optimum unstructured mesh pattern.

3. Verification and discussion on modeling method

Several examples with various geometries are investigated in this section. In each case study, the results of the present simulation are compared with the available experimental and/or theoretical results. Finally, the appropriate procedure for modeling the impact on composite structures is suggested. The numerical simulation is performed on a computer with 2.66 GHz Intel® Core™ 2 Due CPU, and 4.00 GB RAM.

3.1. Impact on composite plate

A plate with the dimensions of $200 \times 200\ \mathrm{mm}^2$ made of T300/934 graphite/epoxy composite with laminate configuration $[0/90/0/90/0]_s$ is considered. The material properties assumed for T300/934 plies are [12]:

$$E_{11} = 141.2\ \mathrm{GPa},\ E_{22} = 9.72\ \mathrm{GPa},\ \nu_{12} = 0.30$$

$$G_{12} = 5.53\ \mathrm{GPa},\ G_{13} = 5.53\ \mathrm{GPa},\ G_{23} = 3.74\ \mathrm{GPa}$$

The materials properties E_{33}, ν_{13} and ν_{23} are not taken into account in the ABAQUS structural analysis, when using S4R and SC8R elements. Due to the planar theory used for S4R and SC8R elements, only the values of E_{11}, E_{22}, ν_{12}, G_{12}, G_{13} and G_{23} are required to define an orthotropic material [16]. The mass density of the plate is $1.536\ \mathrm{kg/m}^3$ and the ply thickness is 0.269 mm. The plate is subjected to transverse impact by a steel sphere impactor of 12.7 mm diameter with an initial velocity of 3 m/s. In order to model the impactor, ∗RIGID BODY option (second method) is applied. The contact properties are considered to be hard and a penalty contact algorithm is used. Also, ABAQUS/Explicit is used to solve the problem of impact on composite plate.

The present results are compared with those presented by Cairns and Lagace [10], Sun and Chen [11], as well as Ghosh and Sinha [12]. The comparison of the results for the contact force history is shown in Fig. 8. In the modeling of target structure, SC8R elements as well as S4R element are used. Investigating Fig. 8, reveals that using SC8R elements leads to more accurate results in comparison with those reported in other references. When using SC8R elements, the maximum discrepancy in maximum contact

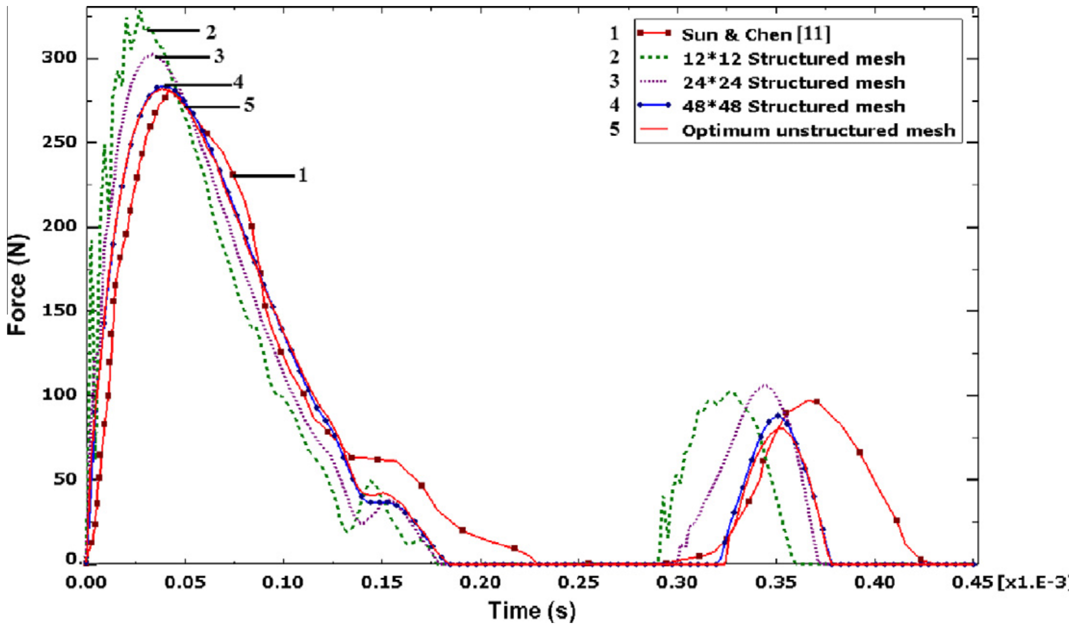

Fig. 6. Effect of number of elements and mesh pattern on contact force history for one quarter of the model.

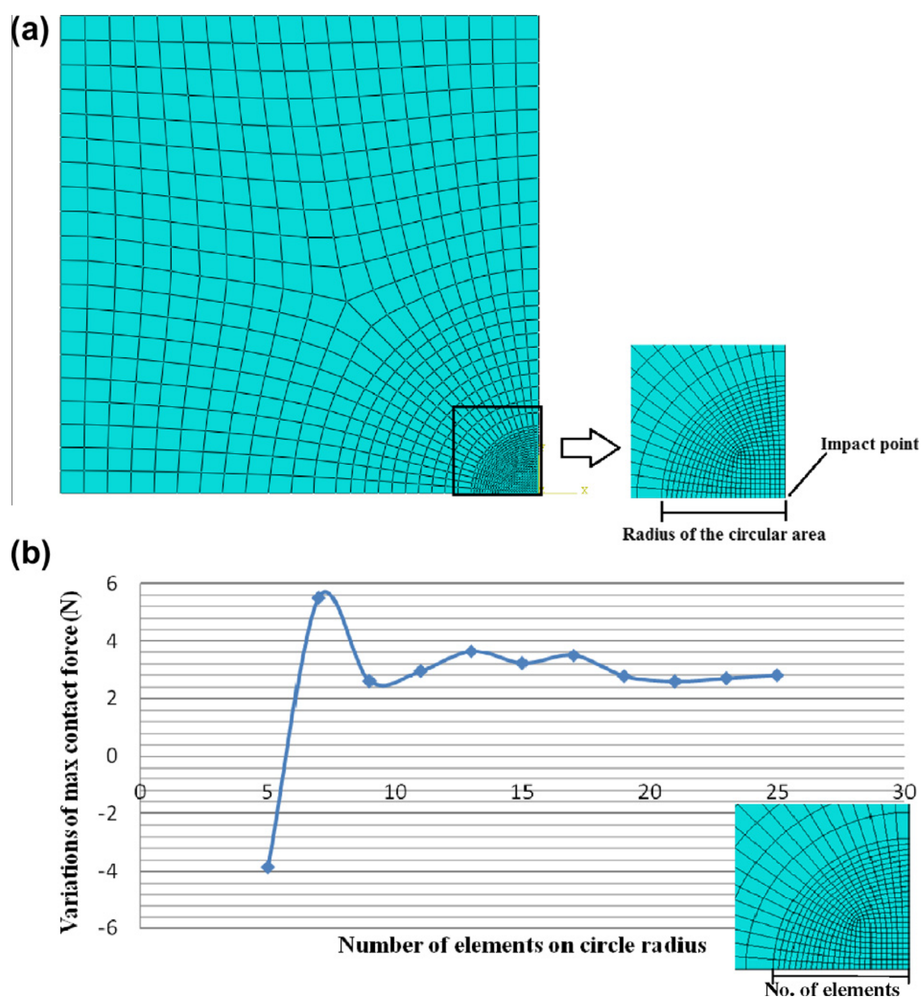

Fig. 7. Convergence study of element numbers: (a) mesh pattern of the circular area surrounding the contact region and (b) maximum contact force versus number of element along the radius of the circular area.

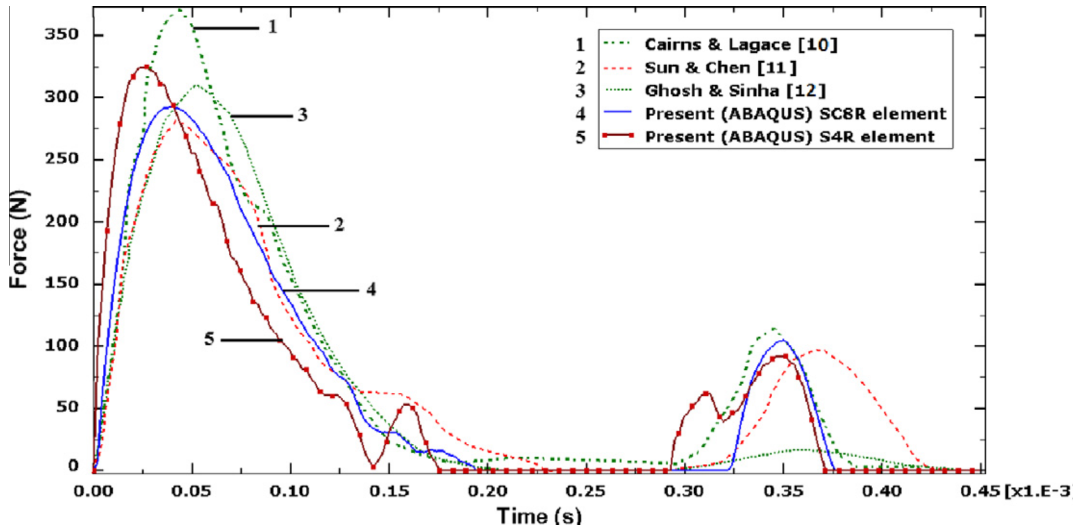

Fig. 8. Numerical and analytical variation of contact force as a function of time for a composite plate.

force is 4.23% and in the time of maximum contact force is 4.05% as compared to more accurate results obtained by Sun and Chen [11]. The discrepancies between the results obtained by the present FE models using SC8R and S4R elements in ABAQUS modeling with those of other references are shown in Table 1.

3.2. Impact on curved composite laminate

In this section, an eight plies panel with a 0.381 m radius, impacted with 0.69 Nm of energy, clamped on the curved edges and simply supported on the straight edges is investigated [13].

Cylindrically curved test specimens with 24.5 cm in axial direction and 12.7 cm in arc length are fabricated with AS4 graphite fiber tape and 3502 epoxy resin [13]. The quasi-isotropic panels have eight plies with an average thickness of approximately 1 mm. The radii of curvature of the specimen is 0.381 m. Fig. 9a shows the effect of solution method on contact force history in curved composite laminate.

The nonlinear FE analysis in the present method, predicts a force history which agrees very well in shape and magnitude with the experimental impact energy of 1.02 Nm, indicating that the actual test velocity may indeed be lower than the value used in the analysis, as indicated by Kistler and Wass [13].

Both SC8R and S4R elements are suitable for modeling the considered curved panel in the example of Kistler and Wass. But, for the sake of time saving, the S4R element is used for modeling the curved panel. First, the impactor is modeled as a deformable body (third method) and then it is modeled as a rigid body (second method). The contact properties are considered to be hard and penalty contact. Also, ABAQUS/Explicit and ABAQUS/Implicit is used to solve the problem of impact on curved composite panel. Fig. 9b shows the effect of method of impactor type modeling on contact force history in curved composite laminate.

In Table 2, the CPU run-times of explicit and implicit solver are compared. According to this table, implicit solver is faster than explicit solver. In addition, modeling the impactor as rigid body decreases the solution time.

3.3. Impact on composite thick cylinder

The material examined in this section is manufactured by the filament winding process with $[\pm 55]_{10}$ stacking sequence lay-up [14]. E-glass fibers are impregnated with a low viscosity epoxy resin. The cylinders used have an internal diameter of 55 mm and a wall thickness of 6.5 mm. Samples for impact and pressure testing are 110 mm long. The tubes are composed of transversely isotropic plies. The characteristics of implemented materials are given in Table 3 with brittle elastic behavior.

A comparison of force versus time using the present model and the numerical and experimental test results of Ref. [14] is provided in Fig. 10a.

The types of elements used for modeling the composite cylinder are S4R and SC8R elements. First, the impactor is modeled as a fully rigid body (first method) in both cases of S4R and SC8R elements in Fig. 10a and b. Then, for the case of SC8R elements, first the impactor is modeled as a fully rigid body (first method) and then as a deformable body (third method) as shown in Fig. 11. The contact properties are considered to be hard and penalty contact. Studying the contact force history shows that assuming the impactor as a rigid body, due to high modulus of steel compared to composite in this case, has the minor effect on the impact dynamics response.

3.4. Impact on composite thin cylinder

The material and geometry of the impactor and the cylindrical shell used in this example is according to the experimental results reported by Swanson et al. [15] and are listed in Table 4. Strain gauges were located on the cylinder according to the sketch indicated in Fig. 12 in which d_1 = 16.76 mm and d_2 = 18.11 mm.

In Fig. 12, A indicates strain gauges in axial direction and H indicates strain gauges in hoop direction. The average strain of the four elements around the node located at the appropriate coordinates on FEA model is calculated. The calculated strains versus time using present model are illustrated in Fig. 13a and b along with the test data taken from Swanson et al. [15].

The type of element used for modeling the composite cylindrical shell is S4R. The impactor is modeled as a fully rigid impactor (first method). The contact properties are considered to be hard and penalty contact. ABAQUS/Explicit is used to solve the problem of impact on the composite cylinder. As it is clear in Fig. 13a and b, good agreement between the results is observed. In Fig. 13a, the maximum discrepancy in maximum strain is 5.78% and in the time of occurrence of maximum strain is 34.77% for A1 axial strain gauge and the maximum discrepancy in maximum strain is 38.16% and in the time of occurrence of maximum strain is −7.59% for A2 axial strain gauge. In Fig. 13b, the maximum discrepancy in maximum strain is 13.17% and in the time of occurrence of maximum strain is 1.39% for H1 hoop strain gauge. Also,

Table 1
Discrepancies in the maximum contact force and the time of maximum contact force between the results obtained by the present FE models using SC8R and S4R elements in ABAQUS modeling with those of other references (illustrated from Fig. 8).

Property	Present modeling discrepancies[a] (using SC8R element)			Present modeling discrepancies[a] (using S4R element)		
	Cairns and Lagace [10]	Sun and Chen [11]	Ghosh and Sinha [12]	Cairns and Lagace [10]	Sun and Chen [11]	Ghosh and Sinha [12]
Maximum contact force	20.94	−4.23	5.37	12.56	−15.29	−4.66
Time of maximum contact force	8.06	4.05	23.05	43.12	40.64	52.39

[a] Discrepancy = ((Reference − Present)/Reference) × 100.

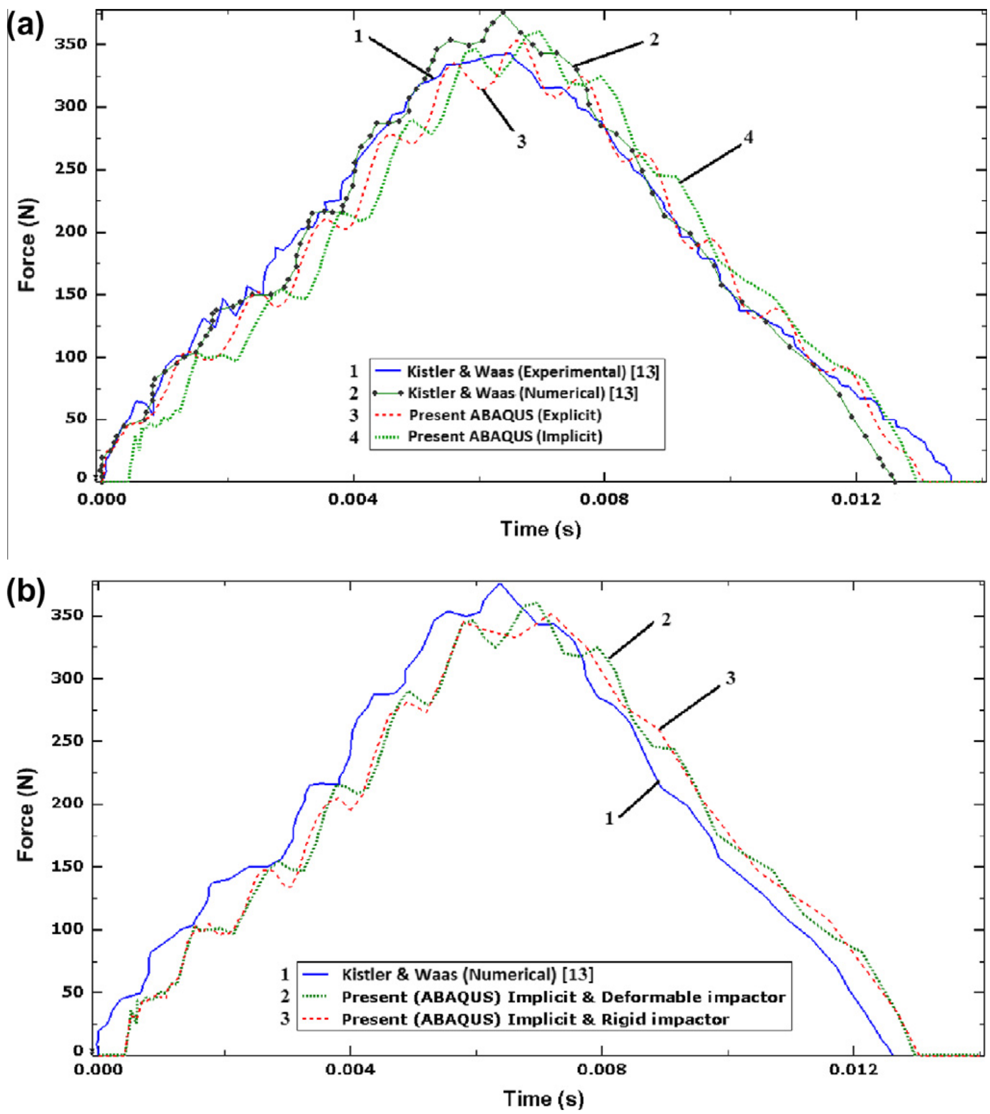

Fig. 9. The contact force history for curved composite laminate: (a) effect of solution method and (b) effect of impactor type.

for H2 hoop strain gauge, the maximum discrepancy in maximum strain is 7.16% and in the time of occurrence of maximum strain is 20.32%.

Table 2
Comparison of CPU run-times (min) for explicit and implicit solver for the curved composite laminate.

Type of solver	CPU run-time (min)
Explicit solver (deformable impactor)	17
Implicit solver (deformable impactor)	4
Implicit solver (rigid impactor)	3.4

The discrepancies especially in the first time steps of the analysis may be mainly due to the inadequate boundary conditions used in the model in comparison with the real experimental configuration. Indeed, the rigid body motion of the whole cylinder under the

Table 3
Material properties for composite thick cylinder with $[\pm 55]_{10}$ lay-up [14].

Material	E_{11} (GPa)	E_{22} (GPa)	G_{12} (GPa)	ν_{12}
Glass/epoxy ply, 0°	49.5	15.9	5.6	0.255

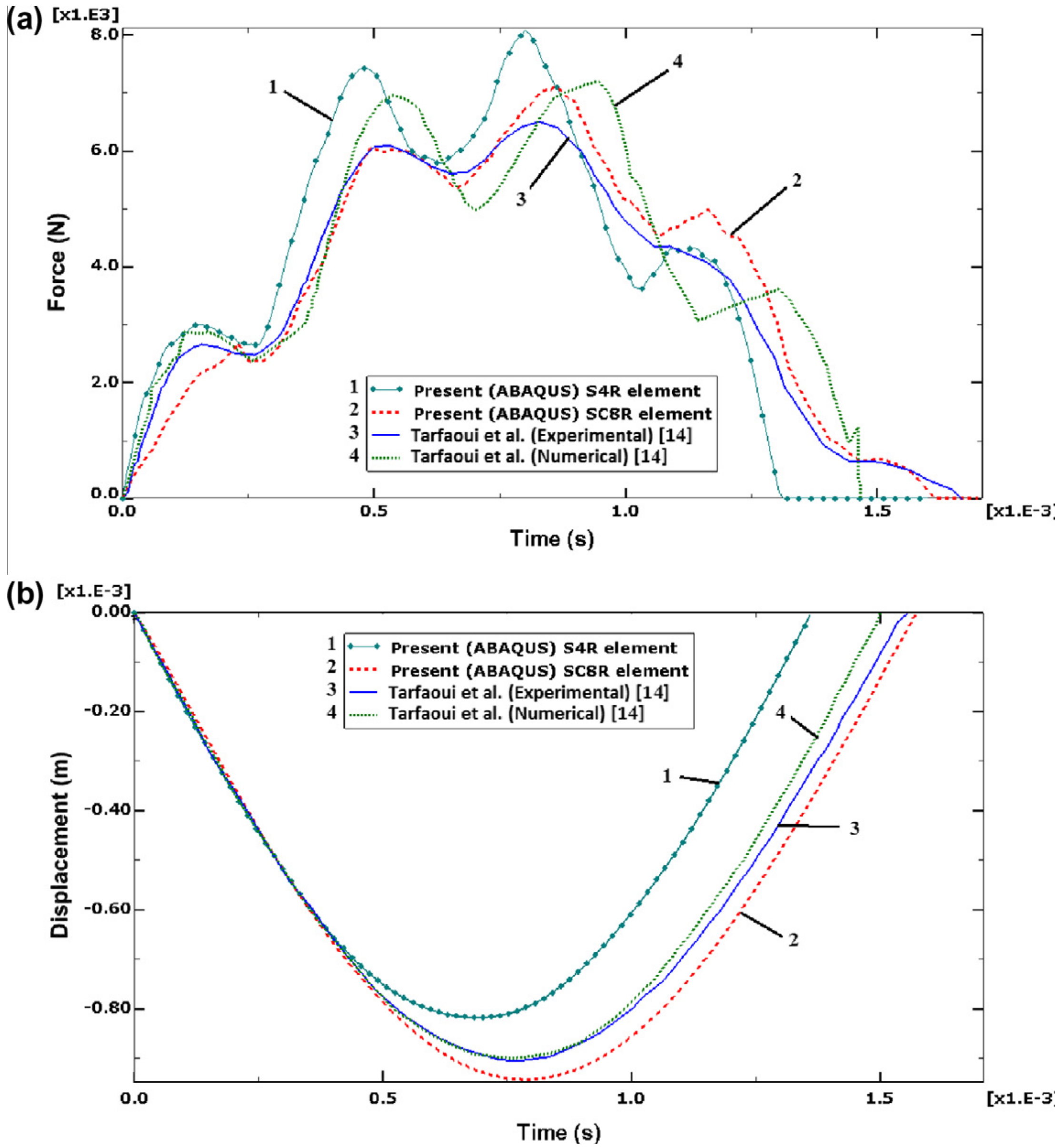

Fig. 10. The comparison of (a) the contact force history and (b) the maximum deflections for composite thick cylinder.

test condition, due to the principle of conservation of linear momentum may influence the results of strain history.

Also, both ABAQUS/Explicit and ABAQUS/Standard are used to solve the problem of impact on thin cylindrical composite shell. Fig. 14 shows the comparison of various solvers in ABAQUS code with experimental results obtained by Ref. [15]. The explicit results show better agreement with the experimental results in Ref. [15]. Using the implicit solver, for H2 hoop strain gauge, the maximum discrepancy in maximum strain is 21.18% and in the time of occurrence of maximum strain is 40.47%.

Comparing the CPU run-time of the explicit and the implicit solver in Table 5 indicates that despite the simulation of the curved composite panel, using explicit solver in this case decreases the time of solution. The large number of elements in this case (Fig. 4) as compared to the curved panel modeling in Fig. 3 is the main reason for this difference.

As indicated in Section 3.2, in the example of Kistler and Waas [13], the implicit solver needs less CPU run-time than the explicit solver to solve the problem of impact on composite curved panel. But, in the example of Swanson et al. [15], the explicit solver needs less CPU run-time than the implicit solver to solve the problem of impact on thin cylindrical composite shell.

4. Discussion

Four types of examples of impact on composite structures are selected. FEM model is made and the solution is done in each example as described in Sections 3.1–3.4. As can be seen in Table 6, these examples include a range of mass of impactor to mass of target structure ratio from 0.01 to 25.6. Also, the selected examples include the composite plates and the cylindrical shells with the side-to-thickness ratios ranging from 4.23 to 114.89. Some examples like that of Tarfaoui et al. [14] have a quasi-static nature. But, the other examples are more dynamic in nature like that of Swanson et al. [15].

4.1. Effect of element type

The discrepancies between the present results and those reported by Sun and Chen [11] and Tarfaoui et al. [14] are shown

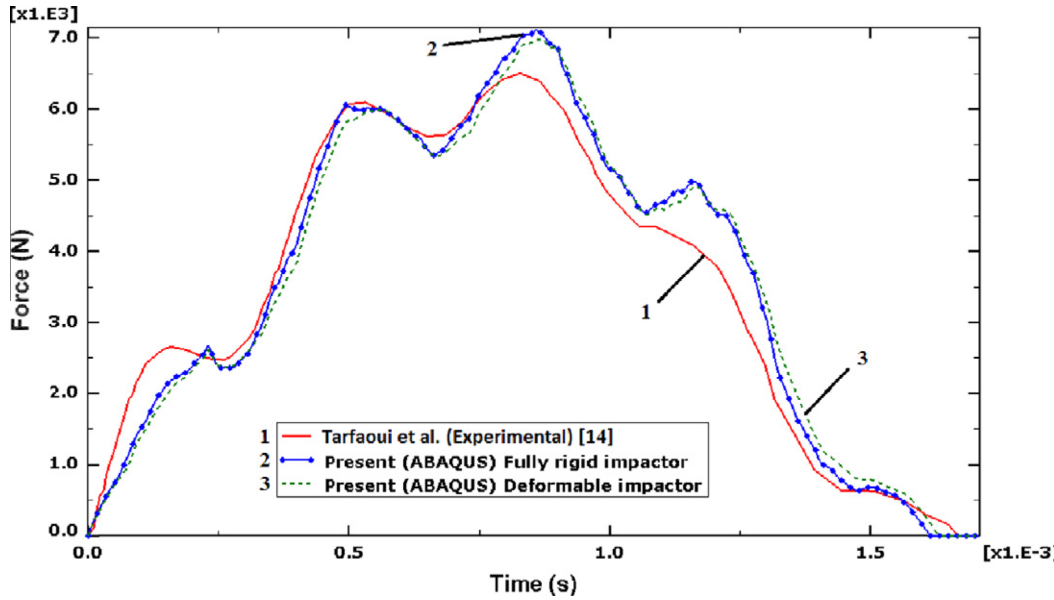

Fig. 11. Effect of types of impactor modeling on contact force history in composite thick cylinder (the example of Tarfaoui et al. [14]).

Table 4
Material and geometry of the impactor and the cylindrical shell [15].

Material	IM7/55A	Lay-up	$[\pm18_2/\pm18_2/90_8]_S$
Longitudinal modulus, E_{11}	148 GPa	Inside diameter	319 mm
Transverse modulus, E_{22}	7.58 GPa	Wall thickness	5.08 mm
In plane shear modulus, G_{12} and G_{13}	3.57 GPa	Cylinder length	1.89 m
Transverse shear modulus, G_{23}	2.62 GPa	Length between supports	0.610 m
Poison ratio, υ_{12}	0.266	Projectile mass	102 g
Poison ratio, υ_{23}	0.311	Projectile tip diameter	20.57 mm

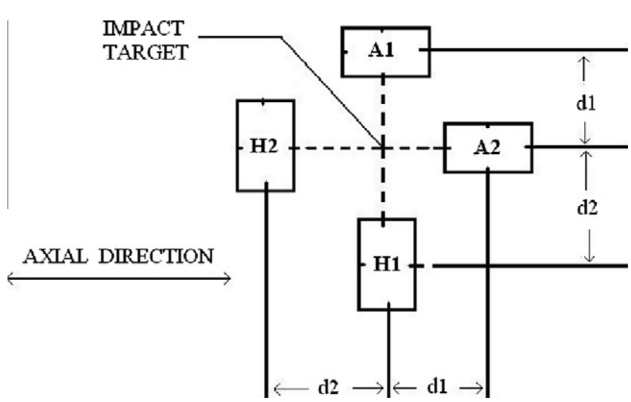

Fig. 12. Location of axial and hoop direction strain gages with respect to impact point [15].

in Table 7. As can be seen in this table, using SC8R in the modeling of composite structures, yields more accurate results compared to the experimental results of Refs. [11] and [14]. Regarding their specifications in these examples, it can be concluded that generally SC8R yields more accurate results compared to the results of S4R. Hence, one may conclude that SC8R is a more suitable element for modeling quasi-static (in the example of Tarfaoui et al. [14]) as well as dynamic impact (in the example of Sun and Chen [11]) problems on both thin and thick composite structures.

As mentioned earlier in Section 2, using SC8R elements in the FEM model, causes the computational cost to be more than that of using S4R elements. According to Table 8, the CPU run-time elapsed to solve the example of Sun and Chen [11] using SC8R is about 1.07 times greater than the CPU run-time elapsed to solve the same problem using S4R. While, the number of degrees of freedom (DOFs) in the FEM model of the example of Sun and Chen [11] using SC8R element is 1.27 times smaller than the number of DOFs in the FEM model of the same example using S4R element. Also, according to Table 8, the CPU run-time elapsed to solve in the example of Tarfaoui et al. [14] using SC8R element is about 38.1 times greater than the CPU run-time elapsed to solve the same problem using S4R element. While, the number of DOFs in the FEM model of the example of Tarfaoui et al. [14] using SC8R element is 10.24 times greater than the number of DOFs in the FEM model of the same example using S4R element. From these comparisons, it can be concluded that, in addition to the number of DOFs in the FEM model, the wave propagation characteristics of the FEM model also influences the CPU run-time. The time increment size for the analysis is equal to the characteristic element dimension divided by the dilatational wave speed of the material [16]. For a special problem, increasing the number of DOFs causes the characteristic element dimension to decrease. Hence the analysis time increment size is decreased and hence the CPU run-time is increased.

4.2. Effect of impactor model

In Table 9, the effect of impactor modeling on the solution time is investigated. As can be observed in this table, the first method which is referred to as fully rigid method has the minimum time consuming in comparison with other methods. In fact, in fully rigid method, the impactor is modeled as a shell with rigid (R3D4) ele-

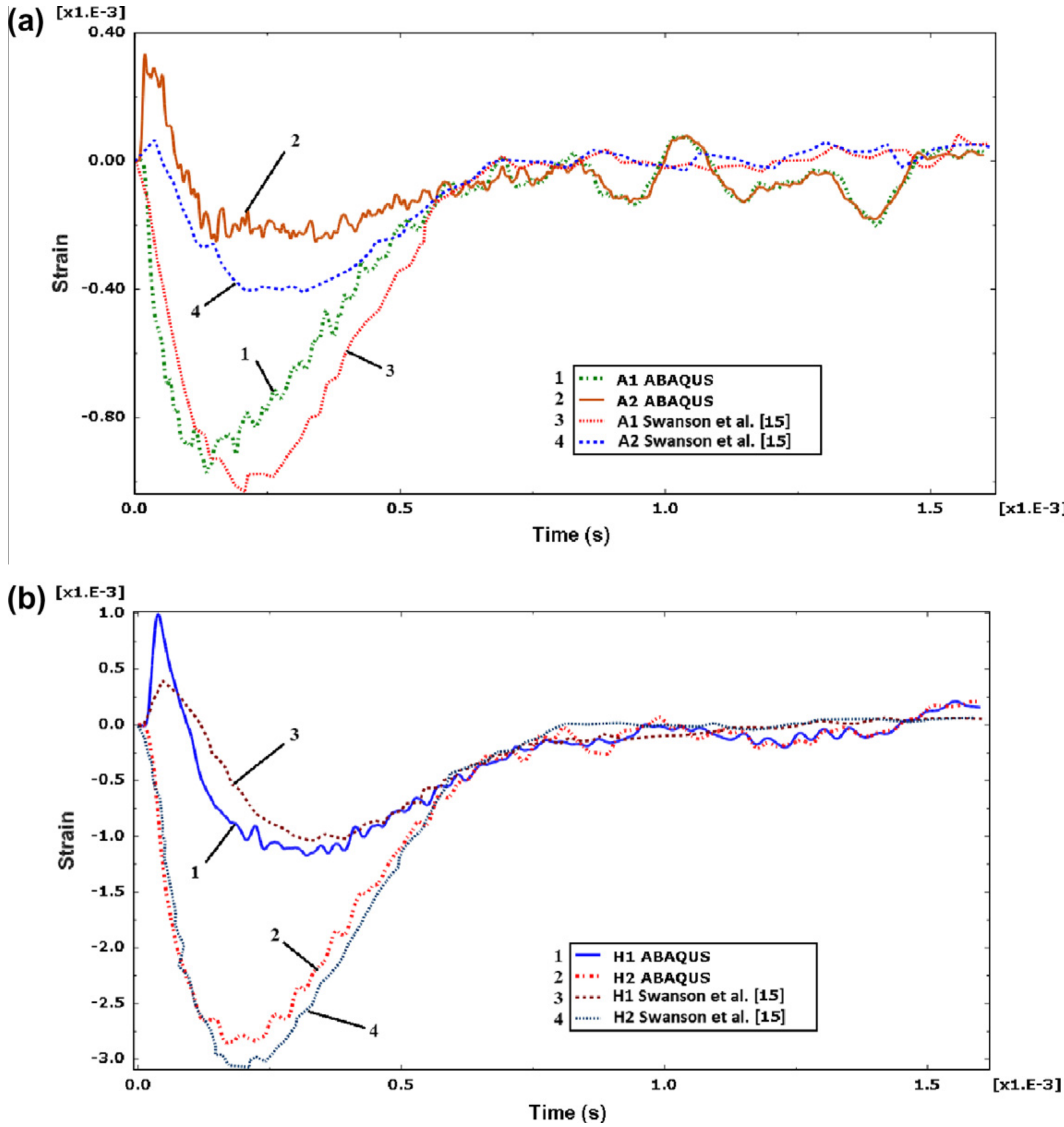

Fig. 13. The comparison of measured [15] and predicted (a) A1 and A2 and (b) H1 and H2 strain gauges responses.

ments. While in the rigid method, the impactor is modeled as a solid sphere with solid (C3D8R) elements which requires less computational effort in comparison with the (R3D4) element. On the other hand, assuming the method of deformable body, leads to the highest CPU run-time among the considered three methods of impactor modeling.

4.3. Effect of solver type

The accuracy of explicit and implicit solvers are compared in Tables 10 and 11, for the examples of Kistler and Waas [13] and Swanson et al. [15], respectively. As can be seen in Table 10, both the maximum contact force ($F_{\max}$) and contact time (CT) are more accurately predicted in the case of the explicit solver. In Table 11, the maximum hoop strain ($\varepsilon_{\max}$) at H2 strain gauge obtained from the methods discussed in Section 3.4 is presented. As it is clear from Table 11, the maximum strain is more accurately predicted using the explicit solver.

For modeling the impact phenomena, the explicit method is preferred. It is suggested for solving dynamic wave-oriented models, like in the cases that the load is applied rapidly and is very severe. This is due to the fact that the governing equations are solved with the full advantage of the computational efficiency of the explicit method and its inherent effectiveness [16,21]. Based on this point and the results shown in Tables 10 and 11, it is more emphasized that the ABAQUS/Explicit solver is a more accurate tool for solving wave-oriented models like the examples considered in this paper.

The CPU run-times for the two examples of Kistler and Waas [13] and Swanson et al. [15] are compared in Table 12. As can be seen in this table, in the example of Kistler and Waas [13] in which the number of DOFs is lower (8574), the CPU run-time used by the explicit solver is 4.25 times greater than that of the implicit solver. But, in the example of Swanson et al. [15] that the number of DOFs is higher (19,812), the CPU run-time used by the implicit solver is 3.17 times greater than that of the explicit solver.

5. Conclusions

In this study, a wide range of low-velocity impact selected examples including a range of side-to-thickness ratios and various impactors with different mass and velocities are investigated. Var-

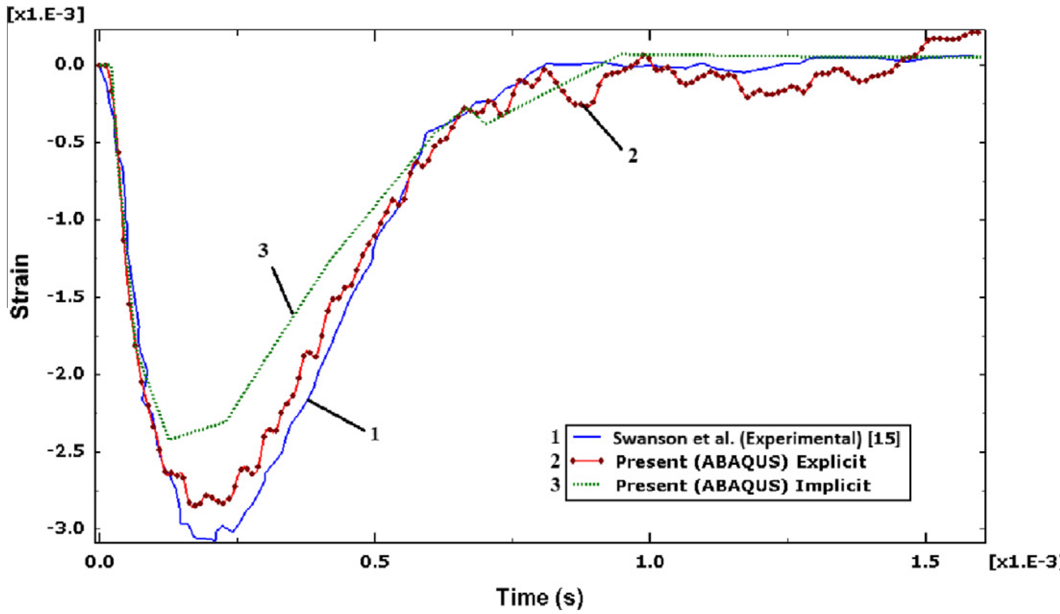

Fig. 14. Comparison of explicit and implicit method in prediction of H2 strain gauge response.

Table 5
Comparison of CPU run-times (min) for explicit and implicit solver in modeling H2 strain gauge.

Type of solver	CPU run-time (min)
Explicit solver	3
Implicit solver	9.5

ious geometries including composite plate, curved composite panel, thin and thick composite cylindrical shell, under quasi-static and dynamic impact, are considered.

In each example, first the results are verified by the experimental and/or theoretical results available in the literature. Then discussions are made in details about the advantages and disadvantages of each of the modeling aspects, including the effect of element type, solution method, impactor modeling, mesh pattern on the impact response of composite structures. Finally, an efficient and appropriate procedure for FEM modeling of impact on composite structures is proposed. The proposed method can serve as a benchmark for impact modeling of composite structures in future investigations.

Table 6
Specifications of the examples considered in the present study.

Example	Mass ratio, m_i/m_t[a]	Side-to-thickness ratio, a/h	Velocity of impact, V_i (m/s)[a]
Sun and Chen [11]	0.15	74.34	3
Kistler and Waas [13]	25.6	114.89	1.097
Tarfaoui et al. [14]	7.24	4.23	1.94
Swanson et al. [15]	0.01	31.40	6.77

[a] Subscripts i and t means the impactor and the target, respectively.

Table 7
Effect of element type on maximum contact force and maximum deflection.

	Type of element present modeling		Sun and Chen [11] (numerical)	Type of element present modeling		Tarfaoui et al. [14] (experimental)
	SC8R	S4R		SC8R	S4R	
F_{max} (N)	293.35 (4.24)[a]	324.45 (15.29)	281.42	7114.56 (9.34)	8080.17 (24.18)	6506.82
w_{max} (mm)	0.39246 (−7.64)	0.38459 (−9.49)	0.42492	0.9480 (4.18)	0.8222 (−9.65)	0.9100

[a] Parenthesis indicate the percentage discrepancy of the present results compared with the results in the Ref. [11].

Table 8
Effect of type and number of element on CPU run-time (min).

Present modeling	CPU run-time (min)		Number of elements		Number of degrees of freedom (DOFs)	
	SC8R	S4R	SC8R	S4R	SC8R	S4R
Example of Sun and Chen [11]	1.66	1.55	3040	2932	14,049	17,862
Example of Tarfaoui et al. [14]	15.25	0.4	23,256	1484	94,392	9222

Table 9
Effect of impactor modeling on CPU run-time (min) in the present FE analysis.

Present modeling	Type of impactor modeling	CPU run-time (min)	Number of degrees of freedom (DOFs)
Example of Kistler and Waas [13]	Rigid	4	8574
	Deformable	3.4	8574
Example of Tarfaoui et al. [14]	Fully Rigid	14.5	105,498
	Deformable	17.5	109,716

Table 10
Effect of type of solver on accuracy of the results in the example of Kistler and Waas [13].

	Type of solver present modeling		Kistler and Waas [13] (experimental)
	Explicit	Implicit	
F_{max} (N)	353.3982 (3.11)[a]	360.7328 (5.25)	342.739
CT (ms)	13.0907 (−3.10)	13.0772 (−3.20)	13.5095

[a] Parenthesis are indicated the percentage discrepancy of the present results compared with the results in the Ref. [13].

Table 11
Effect of type of solver on accuracy of maximum strain, ε_{max} (μm/m) at H2 strain gauge in the example of Swanson et al. [15].

	Type of solver present modeling		Swanson et al. [15] (experimental)
	Explicit	Implicit	
ε_{max}	−2853.809 (−7.16)[a]	−2422.848 (−21.18)	−3073.9

[a] Parenthesis are indicated the percentage discrepancy of the present results compared with the results in the Ref. [15].

Table 12
Effect of type of solver on CPU run-time (min).

Present modeling	Type of solver	CPU run-time (min)	Number of degrees of freedom (DOFs)
Example of Kistler and Waas [13]	Explicit	17	8574
	Implicit	4	
Example of Swanson et al. [15]	Explicit	3	19,812
	Implicit	9.5	

For the thin plate and shells, the S4R elements had appropriate accuracy although in these cases the SC8R elements can be used which is more accurate, but leads to more CPU run-time. According to the modeling results, for thick plates and shells, SC8R elements should be used.

It is observed that, for simulation of impact problems where the impactor is rigid as compared to the target structure, the fully rigid method could be applied and requires less CPU run-time. While using the third method, i.e. deformable impactor in the considered examples, leads to more realistic results, but requires more CPU run-time. According to the outcomes of the present study, it could be concluded that in the FEM models with large DOFs, the explicit solver leads to more accurate results with less CPU run-time. While, in the FEM models with small DOFs, the explicit solver may lead to more accurate results, but more CPU run-time. Hence, the explicit solver is the strictly recommended option for solving the impact problem.

References

[1] Kumar S, Rao BN, Pradhan B. Effect of impactor parameters and laminate characteristics on impact response and damage in curved composite laminates. J Reinf Plast Compos 2007;26:1273–90.
[2] Her SC, Liang YC. The finite element analysis of composite laminates and shell structures subjected to low velocity impact. Compos Struct 2004;66:277–85.
[3] Kim SJ, Goo NS, Kim TW. The effect of curvature on the dynamic response and impact-induced damage in composite laminates. Compos Sci Technol 1997;51:763–73.
[4] Oguibe CN, Webb DC. Finite-element modeling of the impact response of a laminated composite plate. Compos Sci Technol 1999;59:1913–22.
[5] Vaziri R, Quan X, Olson MD. Impact analysis of laminated composite plates and shells by super finite elements. Int J Impact Eng 1996;18(7–8):765–82.
[6] Maiti DK, Sinha PK. Impact response of doubly curved laminated composite shells using higher-order shear deformation theories. J Reinf Plast Compos 1996;15:575–601.
[7] Mili F, Necib B. Impact behavior of cross-ply laminated composite plates under low velocities. Compos Struct 2001;51:237–44.
[8] Zhao G, Cho Ch. On impact damage of composite shells by a low-velocity projectile. J Compos Mater 2004;38:1231–54.
[9] Lakshminarayana HV, Boukhili R, Gauvin R. Finite element simulation of impact tests of laminated composite plates. Compos Struct 1994;28:47–59.
[10] Cairns DS, Lagace PA. Transient response of graphite/epoxy and Kevlar/epoxy laminates subjected to impact. AIAA J 1989;27(11).
[11] Sun CT, Chen JK. On the impact of initially stressed composite laminates. J Compos Mater 1985;19:490–504.
[12] Ghosh A, Sinha PK. Dynamic and impact response of damaged laminated composite plates. Aircr Eng Aerosp Technol 2004;76(1):29–37.
[13] Kistler LS, Waas AM. On the response of curved laminated panels subjected to transverse impact loads. Int J Solids Struct 1999;36:1311–27.
[14] Tarfaoui M, Gning PB, Hamitouche L. Dynamic response and damage modeling of glass/epoxy tubular structures: numerical investigation. Composites Part A 2008;39:1–12.
[15] Swanson SR, Smith NL, Qian Y. Analytical and experimental strain response in impact of composite cylinders. Compos Struct 1991;18:95–108.
[16] Dassault System's Simulia Corp., The ABAQUS 6.8-1 user's manual, USA; 2008.
[17] Shivakumar KN, Elber W, Illg W. Prediction of impact force and duration due to low-velocity impact on circular composite laminates. Int J Appl Mech 1985;52:674–80.
[18] Ambur DR, Starnes JH, Prasad Jr CB. Influence of transverse-shear and large-deformation effects on the low-speed impact response of laminated composite plates. NASA TM 107753; April 1993.

[19] Chattopadhyay S. Response of elastic plates to impact including the effects of shear deformation. In: Sih GC, editor. Recent advances in engineering science. Bethlehem, PA: Leheigh University Press; 1977. p. 127–38.
[20] Setoodeh AR, Malekzadeh P, Nikbin K. Low velocity impact analysis of laminated composite plates using a 3D elasticity based layerwise FEM. Mater Des 2009;30(9):3795–801.
[21] Larson RA. A novel method for characterizing the impact response of functionally graded plates. PhD thesis, The Graduate Faculty of Engineering and Management Air Force Institute of Technology; 2008.