CHAPTER-4

GEOMETRIC AND FINITE ELEMENT MODELING OF NPP CONTAINMENT AND AIRCRAFT

4.1 GENERAL

The nuclear containment and aircraft are big in size and complex in structure. Therefore modelling, interaction, meshing, and partition of NPP outer containment wall and aircraft is the topic of study. As the body of aircraft and NPP containment wall both is deformable structure, so the aircraft body has been modelled as a three-dimensional deformable shell element. The concrete in NPP containment wall is modelled as three-dimensional deformable solid element whereas the steel reinforcement bar is modelled as three-dimensional deformable truss element. A rigid plate has been taken to evaluate the reaction force time curve of deformable aircraft that can be applied on the containment wall to predict the response. To evaluate the reaction force time curve, Boeing 707-320 aircraft is considered. The modelling and meshing of aircraft and NPP wall have been carried out using ABAQUS/CAE. The mesh convergence study has also been performed to get the exact size of element for more accurate results.

In this chapter, the geometric dimension of Boeing 707-320 aircraft and some real NPP containment structure (BWR, CMR, FBR and TMIR) and their boundary condition has been discussed sequentially. The size, types and number of elements used in containment wall and aircraft has also been discussed.

4.2 AIRCRAFT BOEING 707-320

The commercial aircrafts of 707 series of Boeing were introduced in service in 1958. The Boeing 707-320 is one of the most common variants in 707 series. The study of aircraft crash on NPP containment was started with Boeing 707-320 aircraft so this aircraft is considered as the bench mark in this research field. In the present study, Boeing 707-320 aircraft is considered to validate the results with existing research. A brief specification of Boeing 707-320 aircraft has been given in Table 4.1 and the detailed drawings are shown in Fig. 4.1 (aviastar.org).

Table 4.1: Boeing 707-320 Specification

Gross Weight	152,400 kg
Passenger capacity	141 passengers mixed class and 189 economy
Length	44.35 m
Wing Area	280 m^2
Wingspan	44.42 m
Cruising Speed	885-974 km/h
Take off length	3250m
Landing length	2200m
Fuel capacity	90.16m^3
Fuselage width	3.75m
Tail height	12.8m
Range	9300km

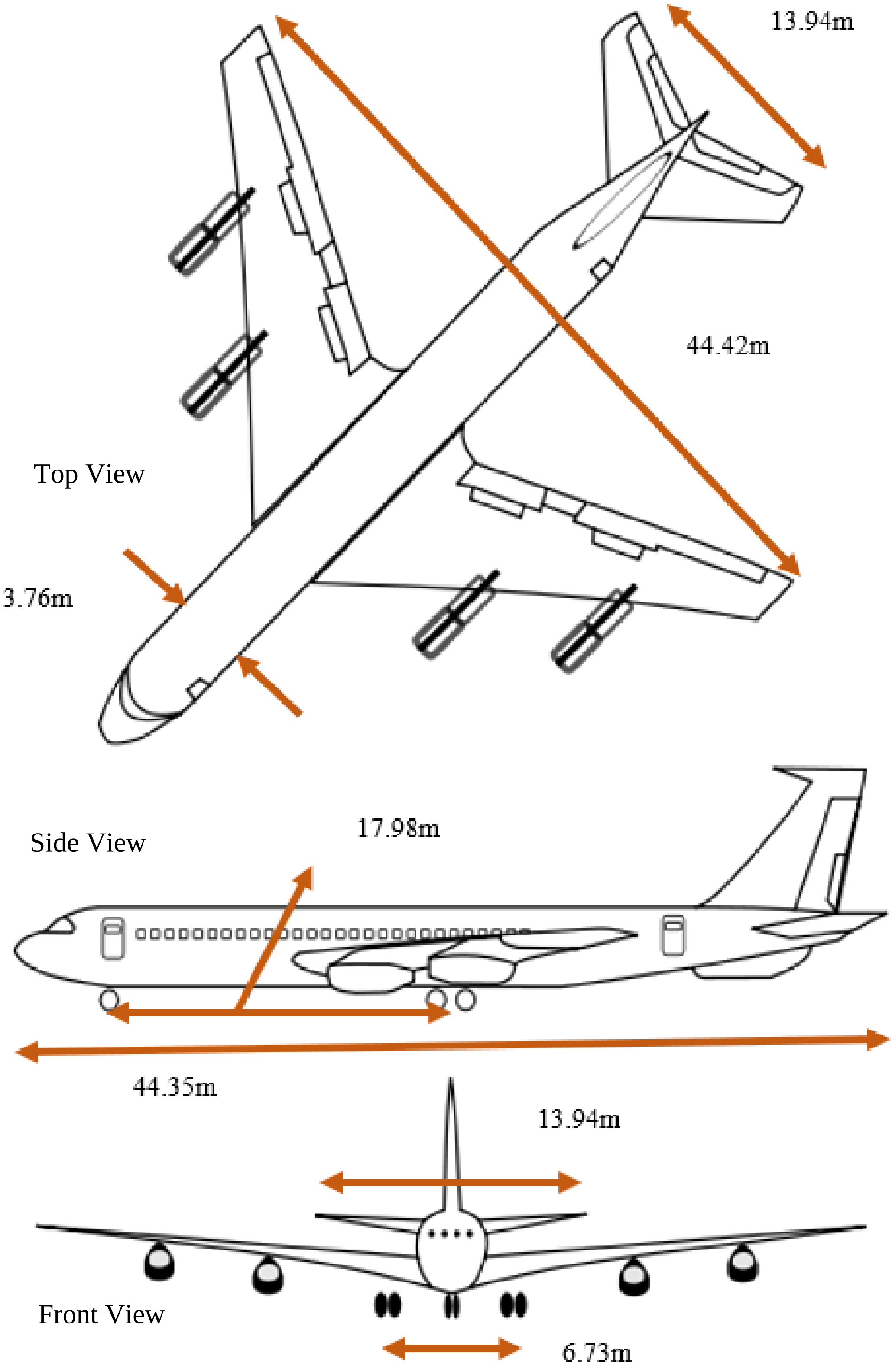

Fig. 4.1 A typical geometrical dimension of Boeing 707-320

4.2.1 Modelling of aircraft Boeing 707-320

The Boeing 707-320 has approximately 45 m length, 45m wing span and two engines at each wing as shown in Fig. 4.2. The fuselage, wings, engines and tails of the aircraft are modeled separately with the help of Fig. 4.1 and Fig. 4.2. Finally, all these parts are assembled at their respective co-ordinates. The aircraft body is carefully partitioned into small parts to enable appropriate meshing during discretization process of the geometry of the aircraft in FEM. The whole body of aircraft has been partitioned into number of parts to overcome the distortion problem in element during simulation which is shown in Fig. 4.3.

4.2.2 Meshing details of Boeing 707-320

The engines and nose of aircraft model have been meshed with a combination of structured and non-structured elements. For the geometric complexity in some portion of the wings and fuselage, these parts can also not be meshed with structured elements. The size and shape of the elements has been decided after properly studying the mesh convergence. In general, the size of element has been considered to be 100mm for all the components of Boeing 707-320 aircraft. A total number of 127562 elements are assigned to the model of aircraft and these elements are linear quadrilateral elements of type S4R (four noded shell element). However, the nose, engines and some other transition regions have been meshed with a combination of S4R and S3R due to nonlinear geometry. The detailed meshing of Boeing 707-320 aircraft is shown in in Fig 4.4.

Fig. 4.2 Real view of Boeing 707-320

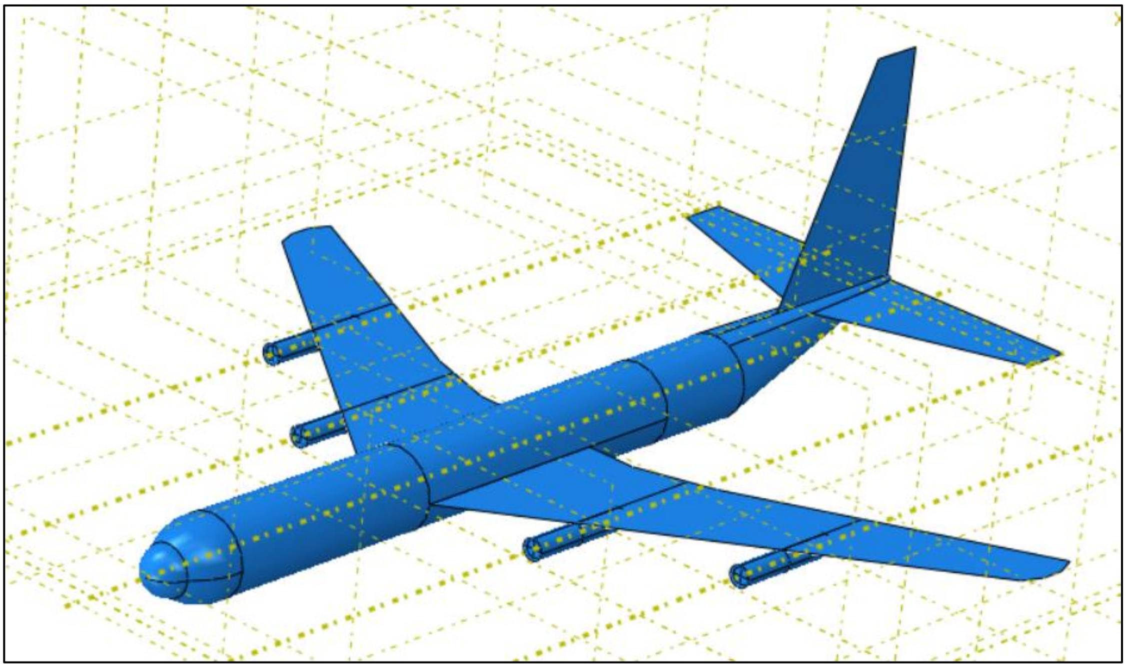

Fig. 4.3 Segmental view of Boeing 707-320 aircraft model for meshing

Fig. 4.4 Meshing detail of Boeing 707-320 in FEM

4.3 MODELING OF HYPOTHETICAL STRUCTURE

In the present study, three hypothtical models are taken into consideration to observe the stuctural behaviour under aircraft crash. The models are:

- PCC flat palte,
- RCC flat plate
- RCC cilindrical wall similar to BWR mark-III containment wall

4.3.1 Modeling of PCC plate

Initially a PCC thick flat plate has been modeled to make the problem simple one. The geometric dimension of plate model is 40 m X 40 m X 3 m which is shown in Fig. 4.5. Maximum plate height and width are 40 m and 40 m respectively. The PCC wall is modeled as 3D deformable solid elements. The plate thickness is constant throughout the plate and the plate structure has fixed supports at the boundary (shown in Fig. 4.5). As the critical location in this case is obvious and that is at the mid height of plate, so impact is applied at 20m height from base of plate.

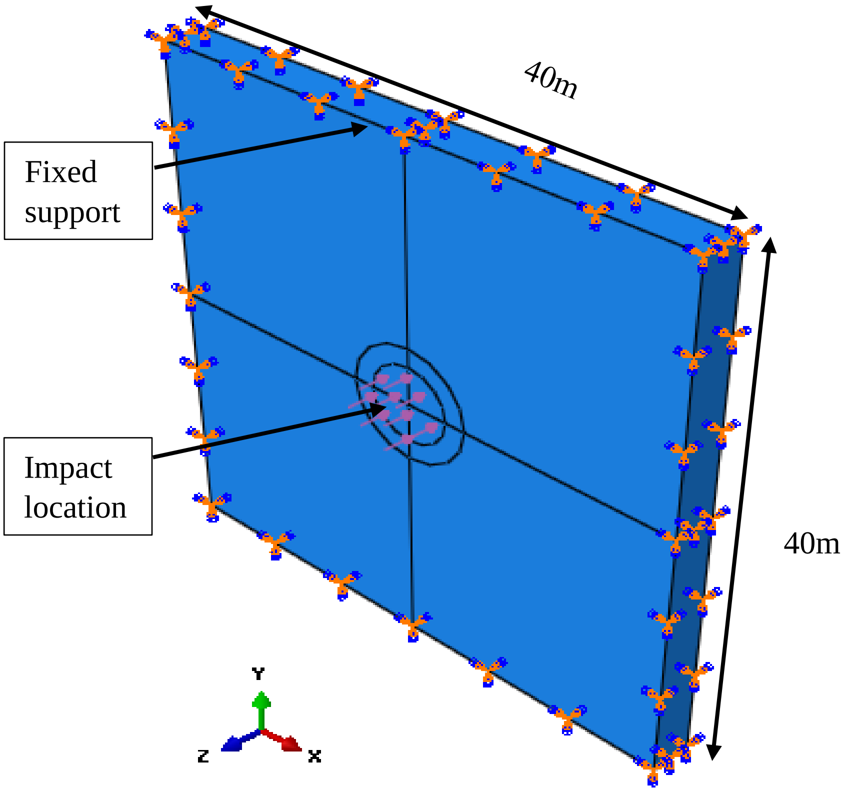

Fig. 4.5 Dimension of PCC plate

4.3.2 Modeling of RCC plate

The geometric dimension of the plate model is 60 m X 60 m X 1.2 m which is shown in Fig. 4.6. The RCC flat plate is modeled as 3D deformable solid elements while the steel reinforcement bar is modeled as 3D deformable wire elements (truss element). The thickness of the RCC plate is kept 1.2m to make a similarity with the thickness of BWR containment wall. The plate thickness 1.2m is constant throughout the plate and the plate structure has fixed supports at the boundary (shown in Fig. 4.6). In this model 40mm dia. reinforcement bar with 80mm centre to centre spacing (approximately 2.5% steel reinforcement) is provided at both the faces inner as well as outer. The 100mm cover is considered in this model both sides. The impact load is applied at the mid height of the plate because the maximum deformation, stress and damage occur in that region.

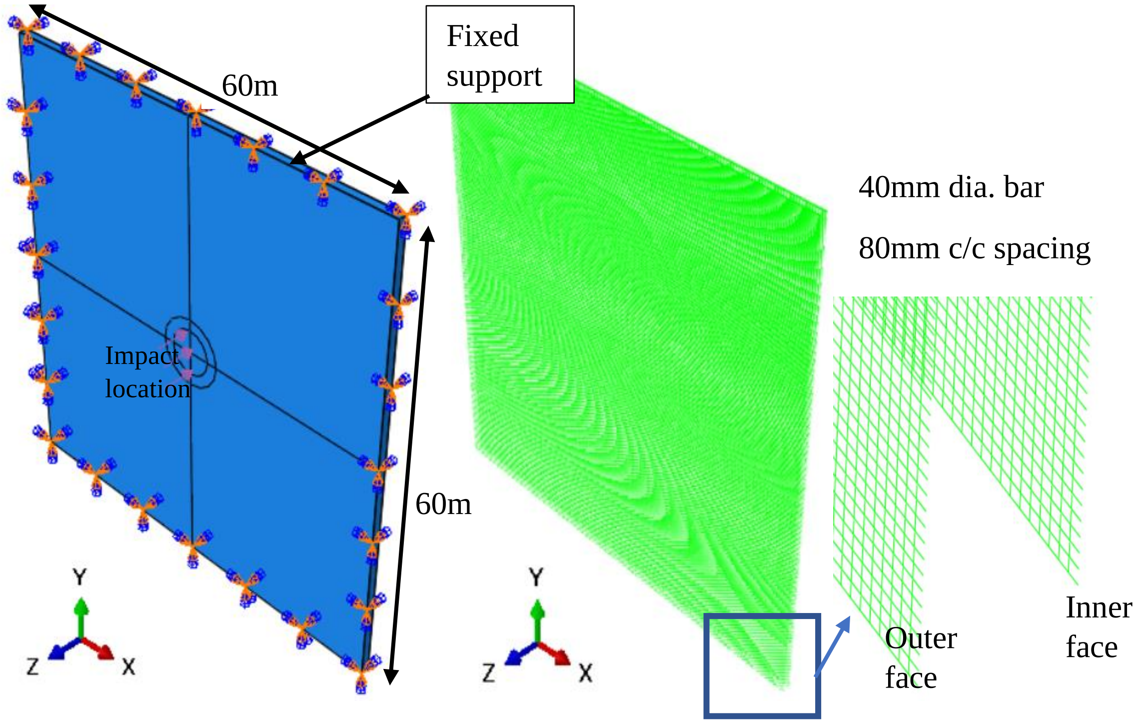

Fig. 4.6 Dimension of RCC plate and reinforcement details

4.3.3 Modeling of RCC cylindrical wall

Before performing on real NPP containment structures, this RCC cylindrical wall has been simulated because maximum NPP containment is cylindrical in shape. The dimension of the model is similar to BWR mark-III containment wall but here fixed support has been used in place of circular dome roof. The dimension of structural model is 42 m X 46 m X 1.2 m shown in Fig. 4.7. The concrete in RCC cylindrical containment wall has been modeled as 3D deformable solid elements while the steel reinforcement bar has been modeled as 3D deformable wire elements (truss element). The containment thickness is constant throughout the height of structure. The model has fixed supports at the boundary of top as well as bottom which has been shown in Fig. 4.7. The reinforcement detailing is same as BWR containment wall. In this model 40mm dia. steel reinforcement bar with 80mm centre to centre spacing is provided with 100mm cover. The impact location is considered the mid height of the wall which is shown in Fig. 4.7.

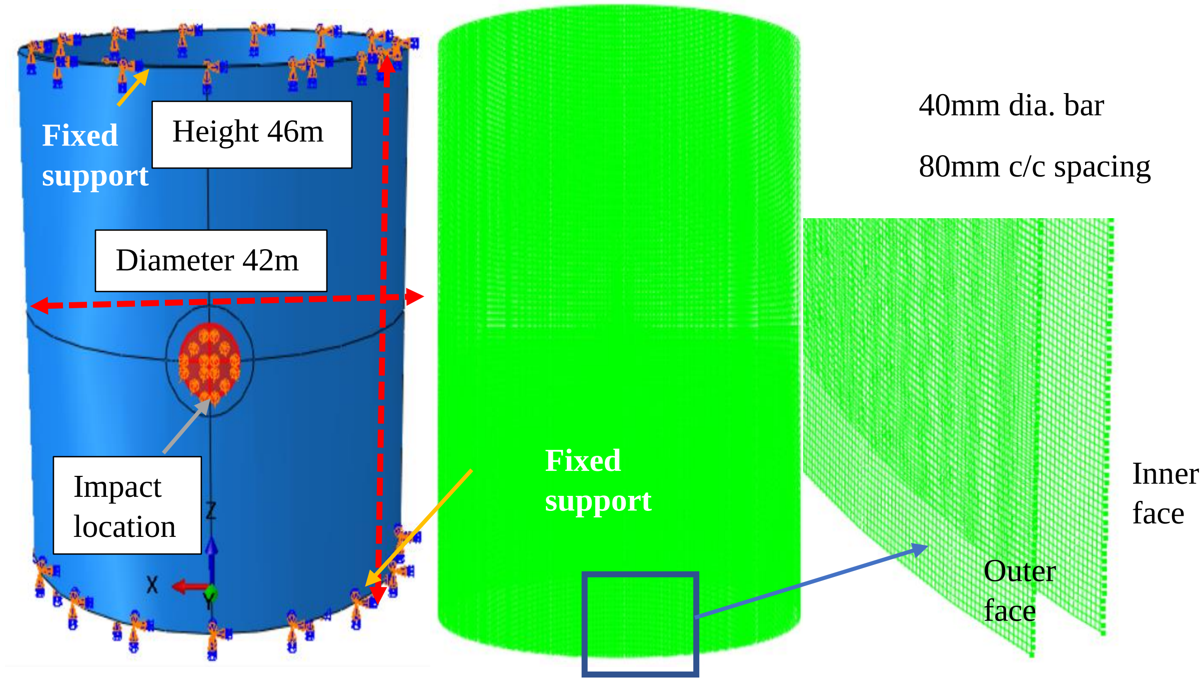

Fig. 4.7 Dimension of RCC cylindical wall and reinforcement details

4.4 MODELING OF SOME REAL NPP CONTAINMENTS

A three-dimensional model of the containment is made using preprocessing module of ABAQUS. In the present study, several analyses are performed to get the behaviour of nuclear containment structure due to aircraft crash. Most of the nuclear containment structure has a cylindrical bottom and circular dome. In the present sudy, four real nuclear containment walls have been considered.

- Boiling Water Reactor (BWR Mark-III)
- The Creys-Malville Reactor (CMR),

- Fessenheim and Bugey Reactor (FBR)
- Three Mile Island Reactor (TMIR)

The structural dimension of every containment is different in size and shape which has been shown in Fig. 4.8 (Bangash, 1982). The geometric model of the BWR containment has been considered that is identical to Abbas et al. (1996). The wall thickness of CMR, FBR, and BWR is constant throughout the containment from base to top except TMIR. The BWR consists of a cylinder and cicular dome portion of 1.2m thickness without steel liner. The CMR has a cylindrical portion and flat roof of thickness 0.85m with 6mm steel liner. The FBR has a cylindrical portion and curve roof of 0.9m thick with 6mm steel liner. The cylindrical portion has 2.5m thickness and dome portion has 1.5 m thickness in TMIR containment. The TMIR structure has 6mm thick steel liner which is provided at the inner face of containment wall. The steel liner of 6mm thickness has been provided at the inner face of containment in every reactor except BWR. TMIR has more height whereas CMR has less in height among the considering NPPs. It is assumed that the containment structure has fixed support at the base for all NPPs. In these model 40mm dia. rebar with 80mm centre to centre spacing is provided (both the faces inner and outer). The containment of the NPP is planned as 3D solid elements (deformable) while the rebar is modeled as 3D wire elements (deformable).

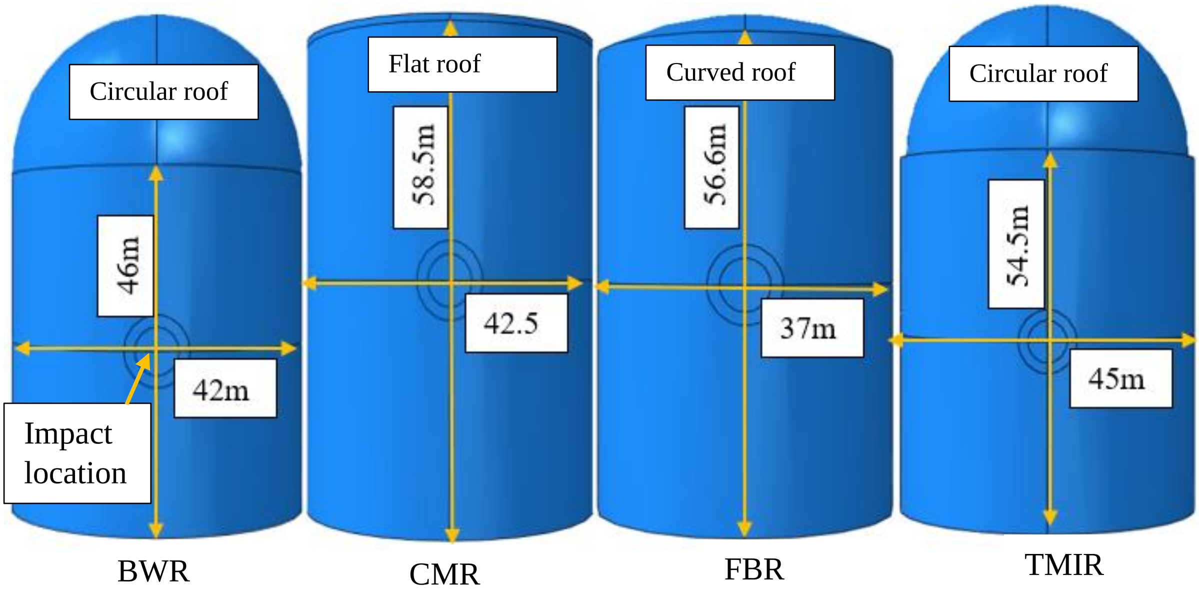

Fig. 4.8 Geometry and dimension of NPPs

The effective cover to concrete is assumed to be 100 mm. The steel reinforcement wire has been modelled in the structure using linear/radial pattern option available in ABAQUS/CAE. The detailing of reinforcement in both faces of the containment structure is shown in Fig. 4.9 for TMIR containment only.

The contact between the concrete and the reinforcement is modeled using embedded element technique available in the ABAQUS finite element code. In this technique the concrete body is used as host region whereas steel reinforcement bar is used as embedded region. The various constraints and interactions have been discussed in detail in section 4.8 of the present chapter.

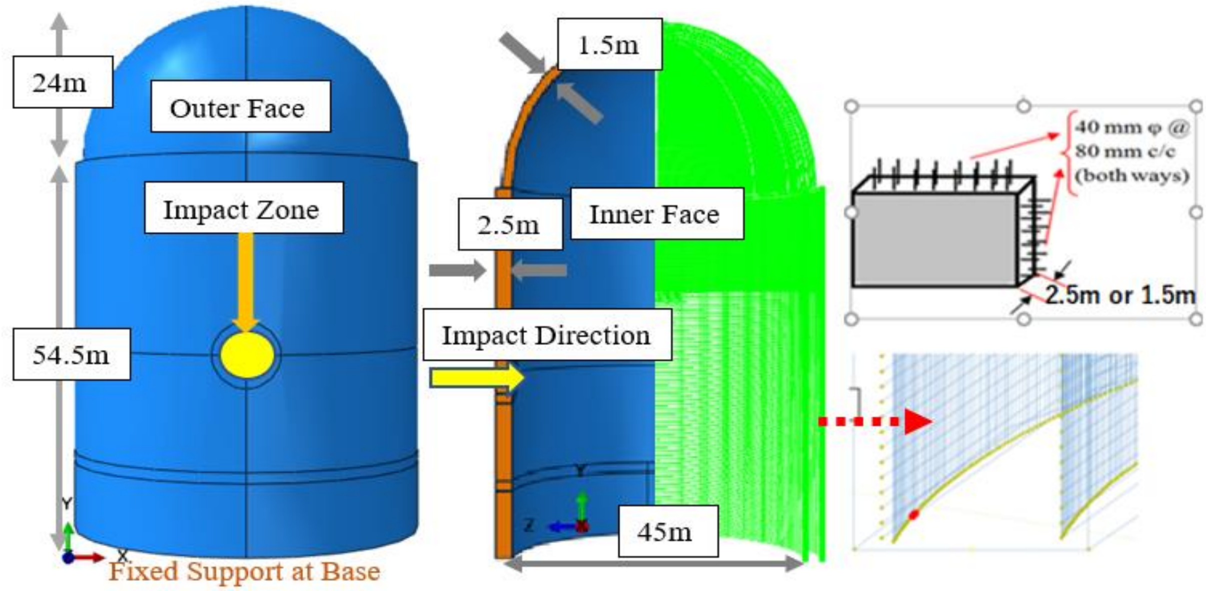

Fig. 4.9 Dimension of Three Mile Island containment and reinforcement details

4.5 MESH CONVERGENCE STUDY

The mesh convergence has been studied for the heat transfer analysis. Before performing the impact and thermal stress analysis on NPP containment structure, a square reinforced concrete wall of size 10 m x 10 m is simulated under fire in order to perform the mesh convergence. The 8-noded brick elements with different size have been taken i.e., 1 m x 1m x 1 m, 0.5 m x 0.5 m x 0.5 m, 0.3 m x 0.3 m x 0.3 m, 0.2 m x 0.2 m x 0.2 m, 0.1 m x 0.1 m, 0.1 m and 0.05 m x 0.05 m x 0.05 m (Fig. 4.10). The temperature variation contours of plates with different size of elements have been shown in Fig. 4.10. The maximum temperature in the containment has been plotted with respect to different size of elements in Fig. 4.11. It is found that the maximum variation in temperature up to 0.25m element size and thereafter the variation is almost constant, indicating no further influence of element size. Hence 0.25m size of elements have been considered for the region under the application of fire in all the simulations of heat transfer and thermal stress analysis. The size of reinforcement elements however, is kept unchanged in every analysis.

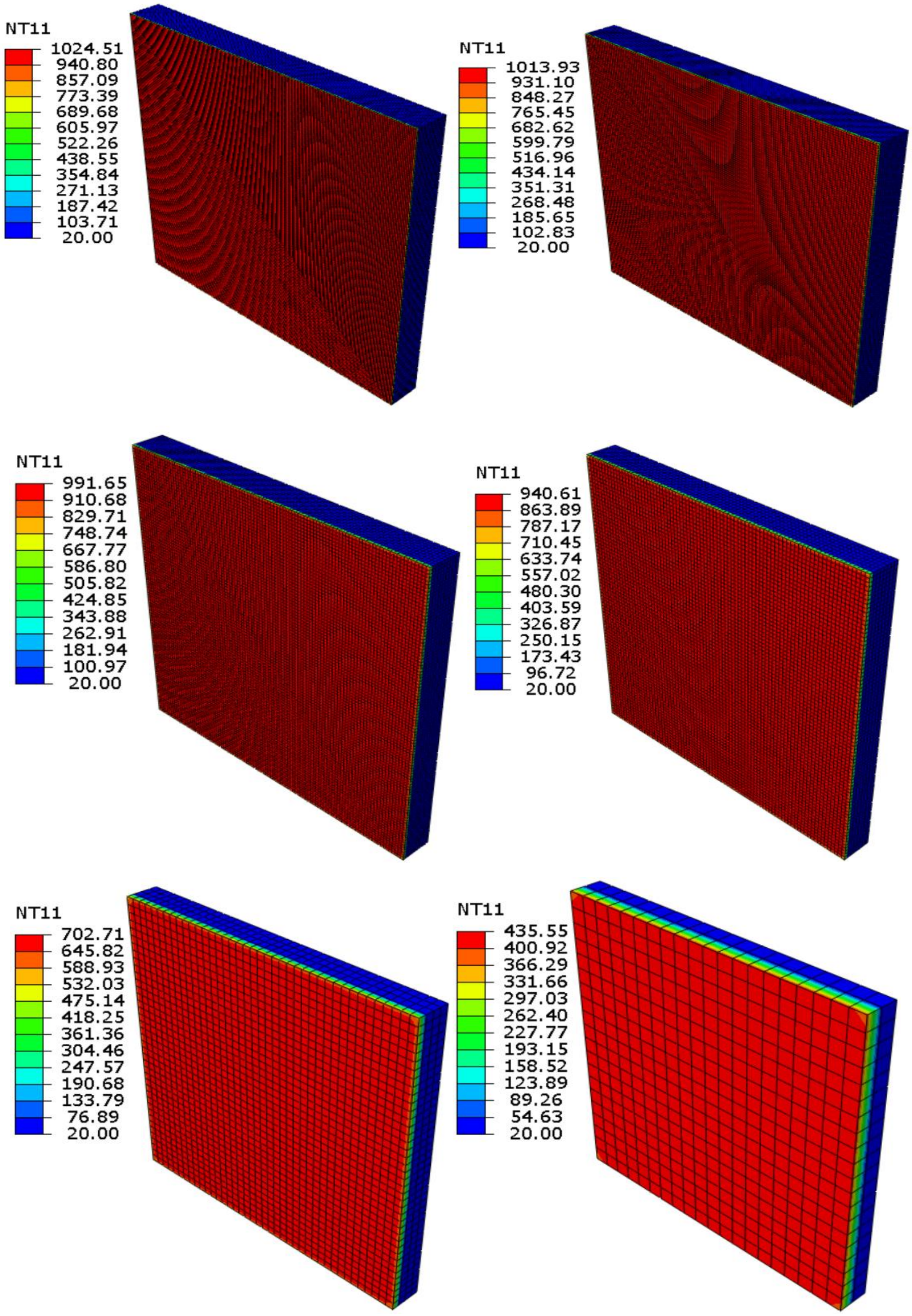

Fig. 4.10 Nodal temperature profile in plate with different size of element

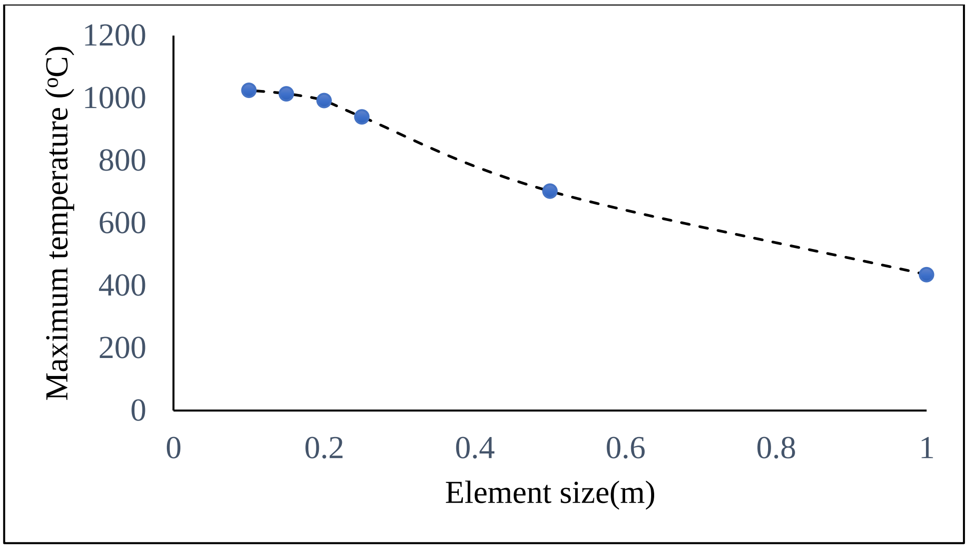

Fig. 4.11 Temperature variation with different size of elements

4.6 MESHING DETAILS OF CONTAINMENT WALL AND REINFORCEMENT BAR

For a suitable meshing, the NPP containment structure is divided into different regions. The meshing is performed carefully to get correct results and to prevent unnecessary distortion of model elements. The detail of meshing for the containment is shown in Fig. 4.12.

In the present discussion meshing details of TMIR containment has been covered only because all models have same size and type of element. But the number of elements is different in every model. The containment has been meshed based on the size of aircraft. Every aircraft has different impact area due to different fuselage span. The central circular region (impact region) of the containment subjected to aircraft loading is meshed with three-dimensional, reduced integration, 8 node brick elements. The application of load for impact on structure is shown in Fig. 4.12 and this is due to the maximum deformation observed in that location. The impact region and 10m lower most part of containment has meshed with 3D-8 noded brick elements (C3D8R) with reduced integration of size 250mm that give 10 elements at containment thickness. The size of elements in the outer area has been changed to 833.33 mm x 833.33 mm x 833.33 mm with 3D-8 noded brick elements (C3D8R) which give three elements in thickness. Tetrahedral elements (C3D4) of various size (from 250 to 833.33mm) are meshed between the intermediate area of impact and outer part of model which has shown in Fig. 4.12. The reinforcement elements are meshed 1m in size with T3D2 (two noded truss element), shown in Fig. 4.13. Above discussed matter is only applicable for the elements in impact and thermal stress analysis. For heat transfer analysis, size of element is same but type is different (DC1D2 for steel rebar, DC3D8 for concrete and DC3D4 for intermediate region). The steel liner has mashed with 4 noded shell elements of size 250mm, Fig. 4.13.

In the present work, the interaction between the elements of concrete and reinforcement is assigned using the embedded scheme in case of impact and thermal analysis but tie constraint is used in heat transfer analysis. Total number of different elements for Boeing 707-320 impact are shown in Table 4.2. Total number of elements in different models are tabulated in Table 4.3. The features of all selective aircrafts have been given in table 4.4. For aircrafts, the force-time history curves are used to apply the loading over the structure. Fig. 4.14 is showing the force history curve for Boeing 707-320/767-400, Phantom F4 and Airbus-A320. Clearly the reaction force due to impact is optimum when the aircraft's wings come into contact with target.

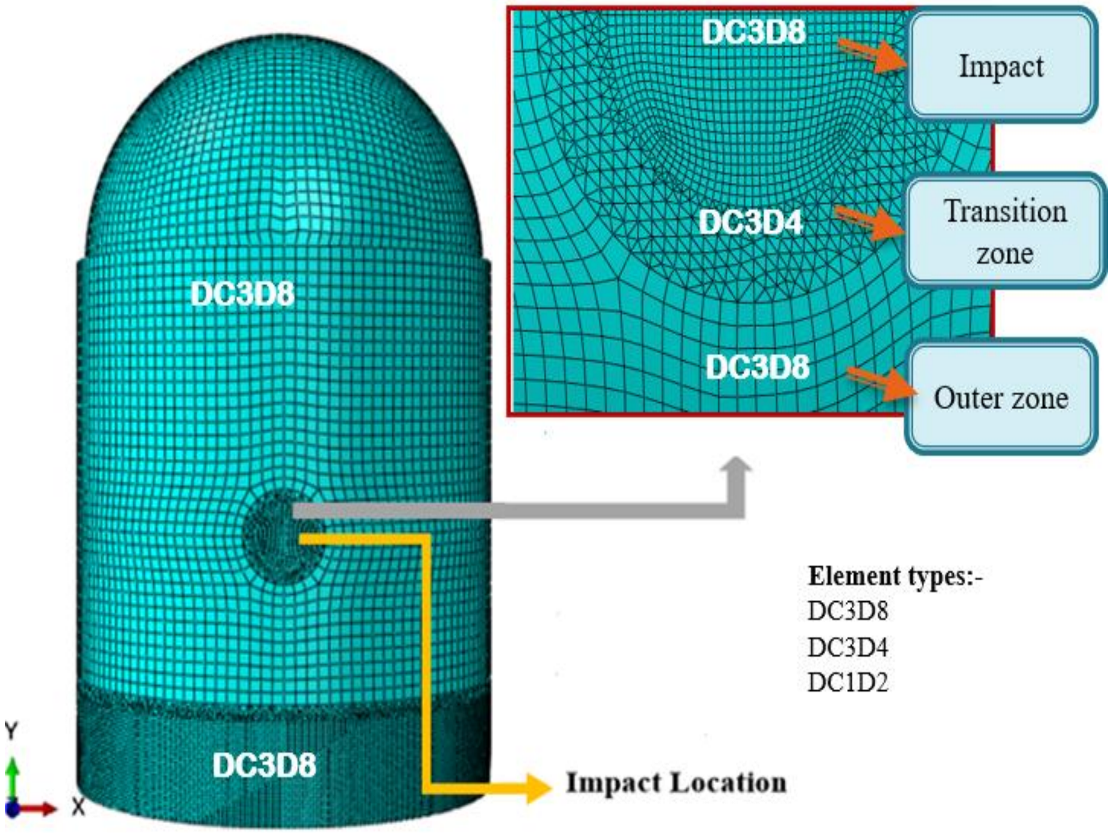

Fig. 4.12 Dimensions, reinforcement, impact location and meshing details

Table 4.2 Number and types of element in aircraft model

Elements type	Number of elements
S4R	114645
S3R	1036

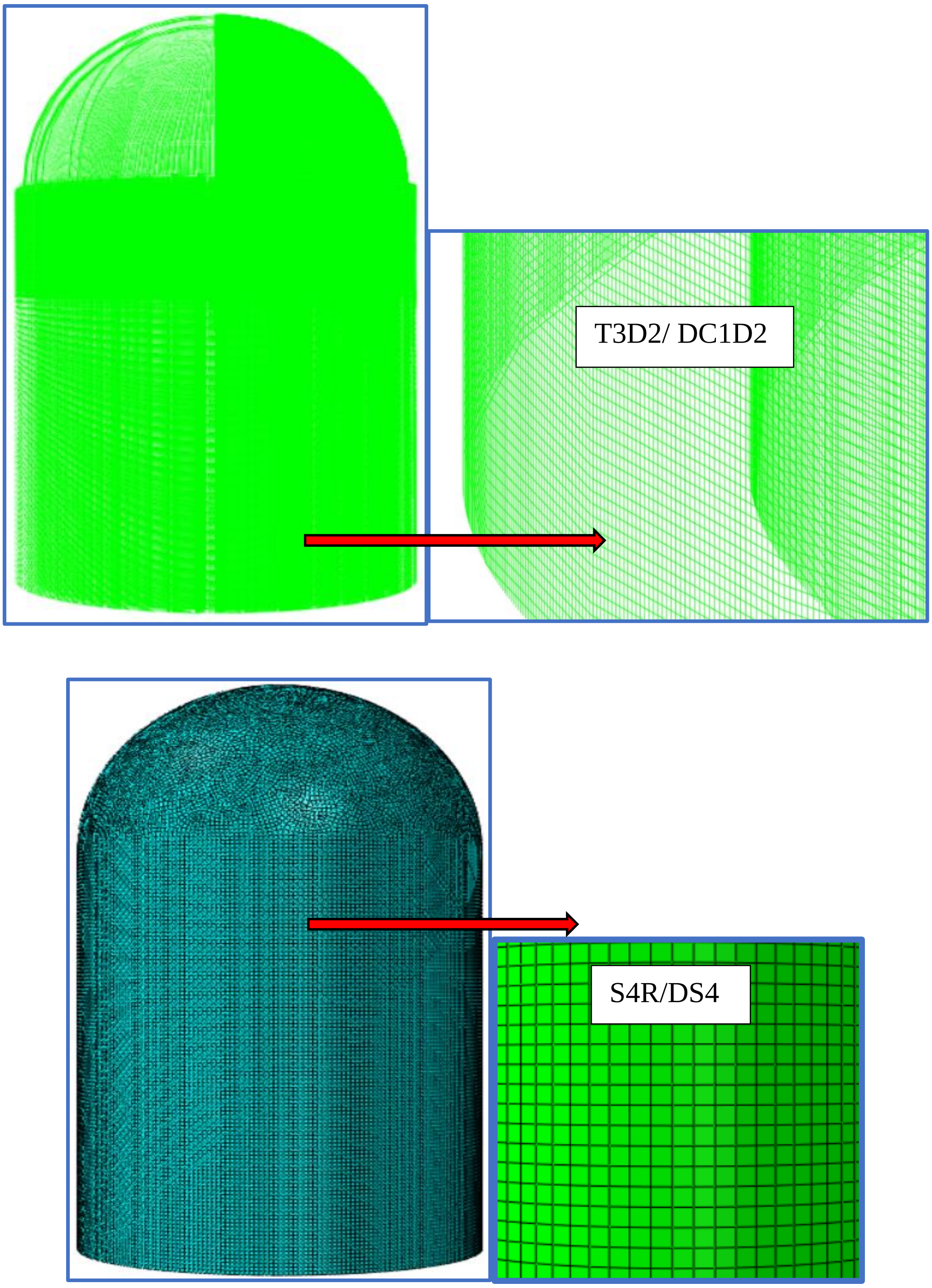

Fig. 4.13 Meshing of reinforcement steel bar and inner steel liner

Table 4.3 Total number of elements in different models

Type of model	DC3D8/ DC3D8	DC3D4/ DC3D4	T3D2/ DC1D2	S4R/DS4
PCC plate	38528	11682	-----	-----
RCC plate	25472	6923	180000	-----
Cylindrical wall	42523	16933	300261	-----
BWR	63598	16930	479630	-----
TMIR	109626	27632	506876	181849
FBR	58692	17251	382012	159620
CMR	65234	21574	446895	180742

Table 4.4. Specifications of different aircrafts

Name of Aircrafts	Length of Aircraft (m)	Radius of Fuselage (m)	Velocity of Impact (m/s)	Peak Impact Force (MN)	Total Time of Impact (Seconds)	Wingspan (m)
Boeing 707-320	44.35	3	103	90	0.3472	44.42
Boeing 767-400	48.5	5.03	150	250	0.4	47.5
Airbus A320	37.57	3.95	120	85	0.3	34.10
Phantom F4	17.74	2.42	215	158	0.08	11.77

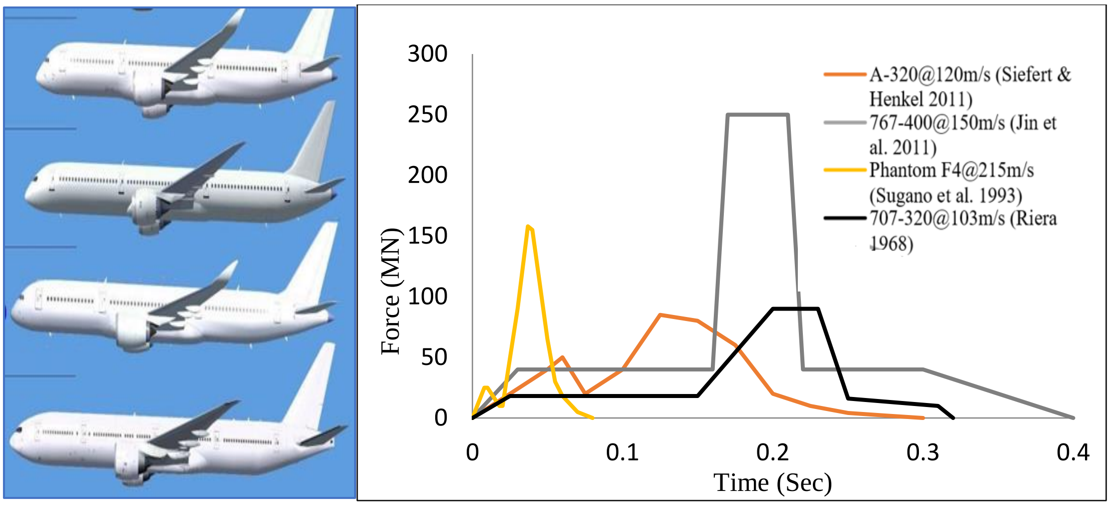

Fig. 4.14 Force-time history curves of different aircrafts

Initially the geometric model of aircraft is applied on the containment for impact analysis, hence the application area has been chosen according to the size of aircraft. There is major three approaches to apply the impact load on NPP containment i.e., reaction force time response curve, geometric model and trifurcation approach.

In the first method, for the application of the curve, an average of the total aircraft contact area is considered. In the trifurcation method the impact area is divided into three regions i.e., fuselage area, first set of engines and second set of engines. Three different reaction time curves are applied on each part. The containment response obtained from the trifurcated and average surface area was then correlated with that obtained from aircraft geometric model approach. Hence, the meshing of the containment has been carried out in three different manners which has been shown in Fig. 4.15. The Table 4.5 showing the number of elements in three approaches. In this case TMIR containment wall has been taken into consideration.

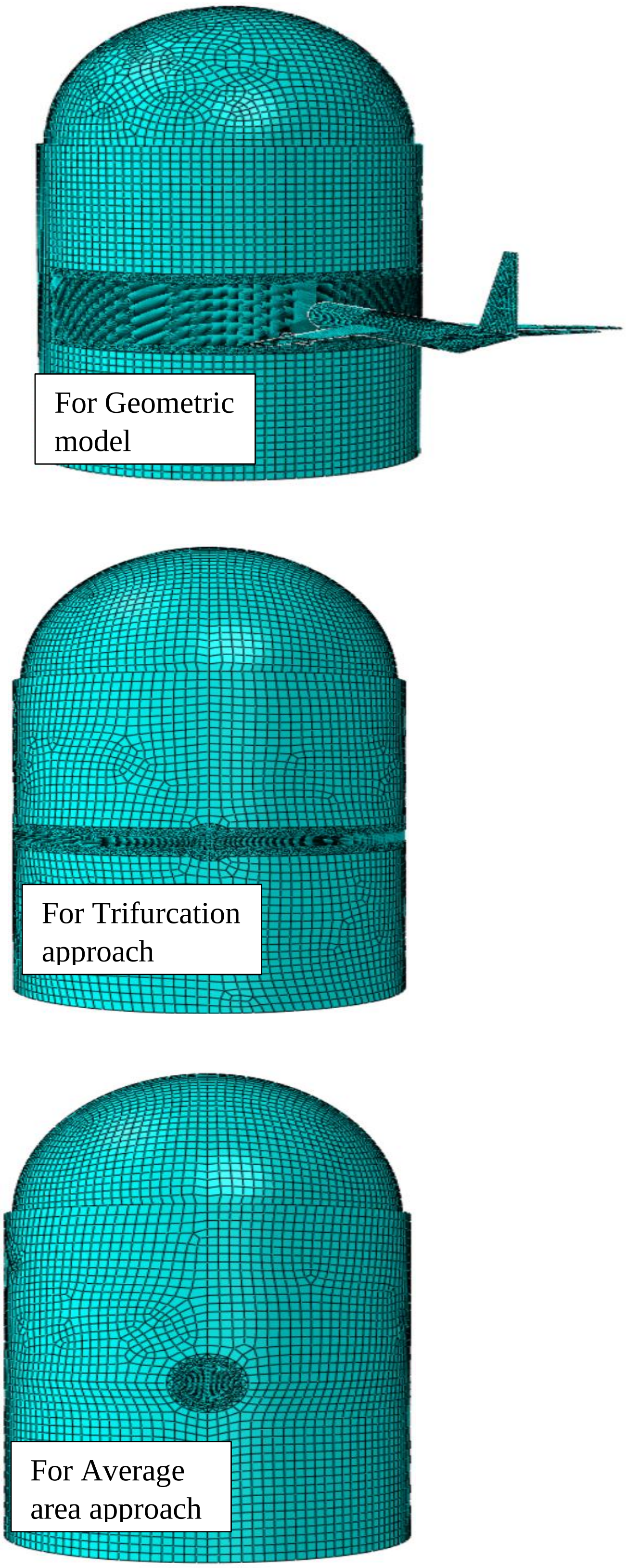

Fig. 4.15 Discretization of the NPP containment for different approach

Table 4.5 Number of elements in containment in different approach

Aircraft	Geometric model	Trifurcation approach	Average area approach
Boeing 707-320	215632	196320	172039

4.7 INTERACTIONS AND BOUNDARY CONDITIONS

ABAQUS does not recognize mechanical contact between the interacting bodies unless the appropriate contact definition has been specified. So, it is not easy to indicate any type of interaction between the surfaces. Here two types of interaction have been discussed.

4.7.1 When the load curve is applied on NPP wall

In the present study the interaction between the elements of the reinforcement and elements of the concrete has been assigned through embedded constraint option for impact and thermal stress analysis. The reinforcement is considered as the embedded and the concrete as the host element. In the embedded element technique, the translational degrees of freedom of the embedded nodes are governed by the degree of freedom of the nearest node of the host element i.e., the reinforcing steel is assumed to have perfect bonding with concrete.

However, embedded constraint is not employed for heat transfer analysis because this constraint does not give accurate results in heat transfer analysis. A surface-based tie constraint is used in heat transfer and thermal stress simulation. The tie constraint works on master-slave formulation and allows to fuse together two regions even though the meshes on the surfaces of these regions may be dissimilar. It constrains each of the nodes on the slave surface to have the same motion and the same value of temperature as the point on the master surface to which it is closest. The constraint prevents slave nodes from separating or sliding relative to the master surface.

If this node is located outside the specified geometric tolerance zone, an error message will be issued. The Geometric tolerance is defined as a limit that how far embedded/slave node can lie outside the regions of the host/master elements in the model. The values of absolute and fractional tolerance have been modified according to the requirements of simulations.

4.7.2 When geometric model of aircraft is appied on NPP wall

ABAQUS/Explicit provides two algorithms for modeling contact and interaction problems i.e., the general contact algorithm and contact pair algorithm. The general contact algorithm can be used only with three dimensional surfaces. While contact pairs can be formed using a pair of rigid or deformable surfaces or a single deformable surface.

Surface-to-surface contact interactions describe contact between two deformable surfaces or between a deformable and a rigid surface. In the present study a self-contact has been defined for the aircrafts body, with an assumption that during deformation different parts of aircraft may come in contact.

The rigid surface must always be the master surface, whereas a node base surface can be used only as a slave surface. Hence, in present study geometric model of aircraft is assumed as the slave surface. As the two bodies come in contact, the penetrations are detected and the contact constraints are applied according to the constraint enforcement method (kinematic). Penetrations of master nodes into the slave surface can go undetected see Fig. 4.16, unless the mesh on the slave surface is adequately refined.

The balanced master-slave contact constraint is used as shown in Fig 4.17. The balanced approach minimizes the penetration of the contacting bodies and, thus, provides more accurate results. There are two different approaches for the contact pair algorithm in ABAQUS/Explicit, finite sliding and small sliding. The finite sliding is the most general algorithm which allows arbitrary motion of the surfaces forming the contact pair. The small sliding assumes that although the body may undergo large motions, there will be relatively little sliding of one surface over the other. Only the finite sliding approach is available for self-contact or contact involving analytical rigid surfaces.

A contact interaction property can define tangential behaviour (friction and elastic slip) and normal behaviour (hard, soft, or damped contact and separation). In the present study mechanical constraint formulation has been done through Kinematic contact algorithm. The kinematic contact algorithm uses a kinematic predictor/corrector contact algorithm to strictly enforce contact constraints (for example, no penetrations are allowed).

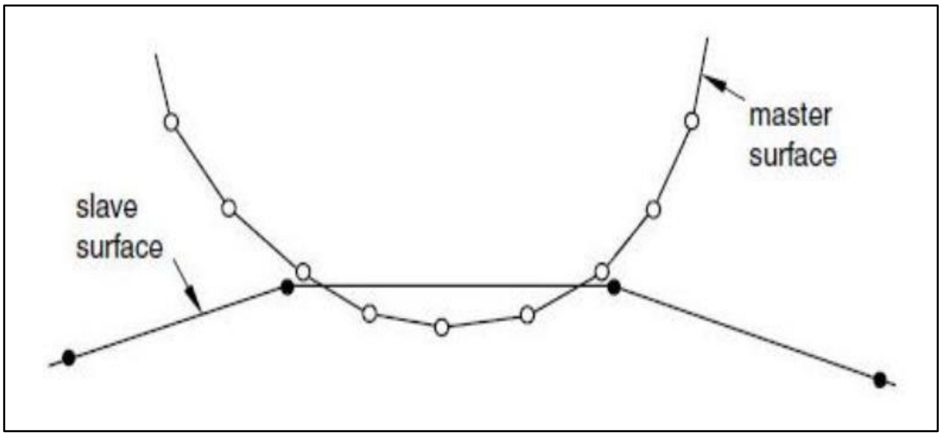

Fig. 4.16 Penetration of master nodes into slave surface with pure master-slave contact

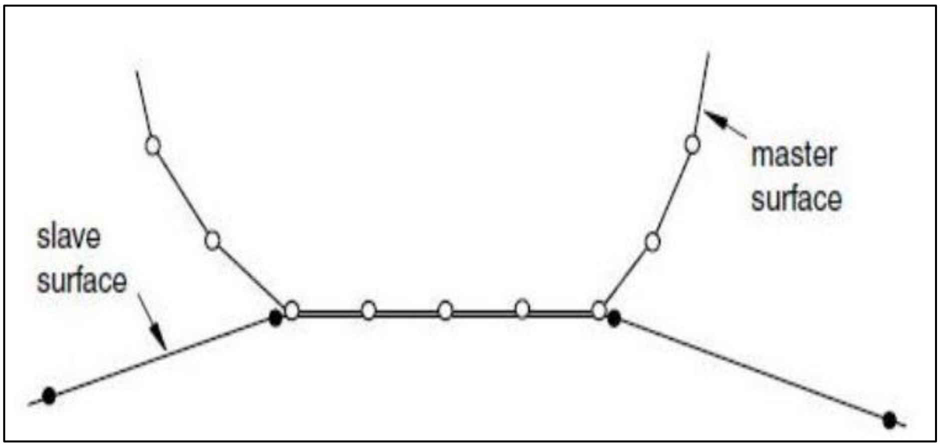

Fig. 4.17 Balanced master-slave contact constraint with kinematic compliance

4.7 SUMMARY

- Discretization of model is very important parameter for numerical simulation.
- The refined mesh should be used at high impact location to get more precise results.
- The model should be properly partitioned to prevent unnecessary distortion of model elements.
- Impact analysis can be done using ABAQUS/explicit scheme and heat and thermal analysis can be done using ABAQUS/implicit scheme. In the present research, the combined impact and thermal effect is there, so implicit scheme must be used.
- To get the most accurate results, the mesh convergence study must be carried out. Because mesh convergence determines how many elements are required in a model to ensure that the results of an analysis are not affected by changing the size of the mesh. System response (stress, deformation) will converge to a repeatable solution with decreasing element size.
- The proper interaction constraints should be used in the model.