DAMAGE ASSESSMENT OF A RECTANGULAR BUILDING AGAINST AIRCRAFT STRIKE

A DISSERTATION

SUBMITTED IN PARTIAL FULFILLMENT OF THE REQUIREMENTS

FOR THE AWARD OF DEGREE

OF

MASTERS OF TECHNOLOGY

IN

STRUCTURAL ENGINEERING

Submitted by:

AYUSH MOHAN
2K18/STE/04

Under the supervision of

DR. RITU RAJ AHIRWAR

(Assistant professor)

DEPARTMENT OF CIVIL ENGINEERING
Delhi Technological University, Delhi.
(Formerly Delhi College of Engineering)

<u>CANDIDATE'S DECLARATION</u>

I, AYUSH MOHAN, Roll No. 2K18/STE/04 student of M.Tech (Structural Engineering), hereby declare that the project Dissertation titled "**DAMAGE ASSESSMENT OF A RECTANGULAR BUILDING AGAINST AIRCRAFT STRIKE**" which is submitted by me to the Department of Civil Engineering, Delhi Technological University, Delhi in partial fulfilment of the requirement for the award of the degree of Master of Technology, is original and not copied from any source without proper citation. This work has not previously formed the basis for the award of and Degree, Diploma Associateship, Fellowship or other similar title or recognition.

Ayush

Place: Delhi AYUSH MOHAN

Date: 31st Aug 2020

CERTIFICATE

I hereby certify that the Project Dissertation titled "**DAMAGE ASSESSMENT OF A RECTANGULAR BUILDING AGAINST AIRCRAFT STRIKE**" which is submitted by Ayush Mohan, 2K18/STE/04 Department of Civil Engineering, Delhi Technological University, Delhi in partial fulfilment of the requirement for the award of the degree of Master of Technology, is a record of the project work carried out by the students under my supervision. To the best of my knowledge this work has not been submitted in part or full for any Degree or Diploma to this University or elsewhere.

Place: Delhi

Date: 31st Aug 2020

Mr. Ritu Raj Ahirwar

SUPERVISOR

Assistant Professor

ACKNOWLEDGEMENT

The success of a MAJOR II project requires help and contribution from numerous individuals and the organization. Writing the report of this project work gives me an opportunity to express my gratitude to everyone who has helped in shaping up the outcome of project.

I express my heartfelt gratitude to my project guide Mr. Ritu Raj Ahirwar for giving me an opportunity to do my Major II project work under his guidance. His constant support and encouragement have made me realize that it is the process of learning which weighs more than the end result. I am highly indebted to the panel faculties during all the progress evaluations for their guidance, constant supervision and for motivating me to complete my work. They helped me throughout by giving new ideas, providing necessary information and pushing me forward to complete the work.

I also reveal my thanks to all my classmates and my family for constant support.

AYUSH MOHAN

ABSTRACT

In the last two decades there has been increase in air traffic along with the density of tall buildings. This has increased the chances of collision, which may be accidental or deliberate. Therefore structures situated near airports and the buildings of high importance must be designed taking this into consideration. Also, the damage assessment of the existing structures needs to be performed against such unpredictable events.

In this thesis, numerical simulations were carried out using ABAQUS/ Explicit software to study the response of a fictitious rectangular steel framed building against collision of Boeing 767-400 ER. The nature of problem is highly complex as a matter of fact that the two interacting bodies demonstrate different mechanical behaviour and damage response due to their distinct stiffness and material properties. Johnson Cooks model have been incorporated to simulate the behaviour of steel, which takes into account high strain rate and increased temperature due to friction between colliding bodies. For concrete, Concrete damage plasticity(CDP) model is used, which follows tensile cracking and compressive crushing failure mechanism.

Different locations of strike of aircraft having weight 204 tonnes and speed 104m/s has been studied. Top corner of the building was found to be most critical point.

CONTENT

LIST OF FIGURES

LIST OF TABLES

CHAPTER 1 INTRODUCTION

Aircraft collision with a building is a low probability-high risk event. Therefore building of high importance must be designed taking this into consideration. Also building situated near airports are always at risk of such accidents. Hence, damage assessment of such structures needs to be performed against such unpredictable events and a careful investigation must be carried out and studied.

In this study, impact of Boeing 767-400 ER at speed of 104 m/s on steel framed structure is studied. ABAQUS finite element software is used for numerical simulation. The nature of problem is highly complex as a matter of fact that the two interacting bodies demonstrate different mechanical behaviour and damage response due to their distinct stiffness and material properties. Johnson Cook model and Concrete Damage Plasticity model has been incorporated to simulate the behaviour of steel and concrete respectively, at high strain rate.

1.1 Steel framed building

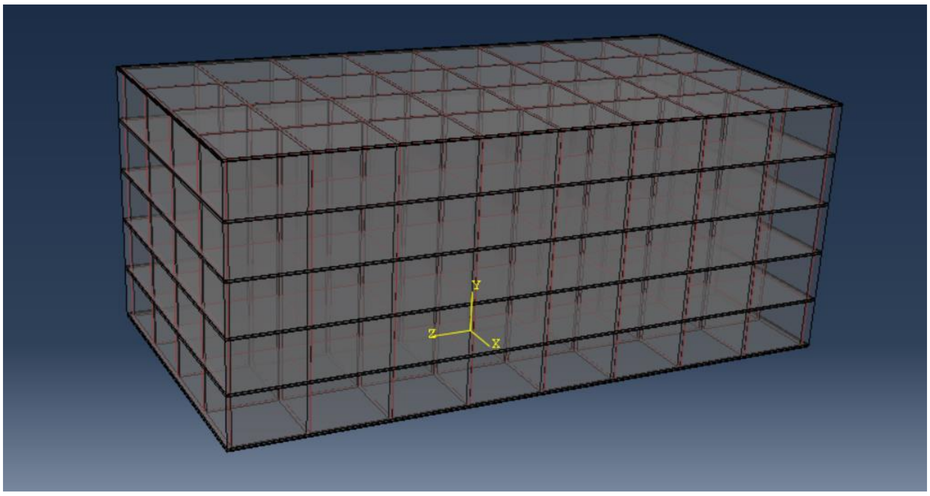


Figure 1.1: steel framed building

For impact, steel framed building is modelled. It is made up of three components:

1. Steel frame-The main frame of the building is made up of 500 MPa grade steel. Beams and columns are made of I section of size 400 mm and 250 mm with web thickness of 9mm. each bay is 6.25 m long along length and 6.5 m along width. Modelling of beam and columns are done using wire elements.
2. RCC slab- Slabs of thickness 20 cm are modelled using solid elements. Material used for them is M35 concrete with suitable reinforcement.
3. Walls- Concrete blocks of M20 grade are used with thickness of 0.5 m in exterior walls and 0.3 m in interior walls.

1.2 Boeing 767-400 ER

In this research, Boeing 767-400ER is used for strike which is an American wide body commercial jet airliner and cargo aircraft. It is manufactured by Boeing's Commercial Airplane Unit in the United States.

Boeing 767 is one of the largest aircraft with a fuselage length of 61.4 m, wingspan of 51.9 m and tail height of 19.4 m. It weighs about 104 tonnes when operating empty weight and its maximum take-off weight is 204 tonnes.

Figure 1.2: Boeing 767

CHAPTER 2 LITERATURE REVIEW

Few researches have been made earlier to assess the damage in Nuclear Containments due to aircraft crash but the impact of crash on a residential building had never been studied.

Earlier researchers have performed uncoupled analysis in which empirical equations were used instead of exact aircraft model. Most of them used Riera (1968) load history curve. Jorge D.Riera [1] assumed aircraft as a soft missile consisting of two zones, a very thin deformation zone in front within control volume S_d and a rigid zone in rear part within control volume S_r. Then he obtained its reaction against a rigid surface.

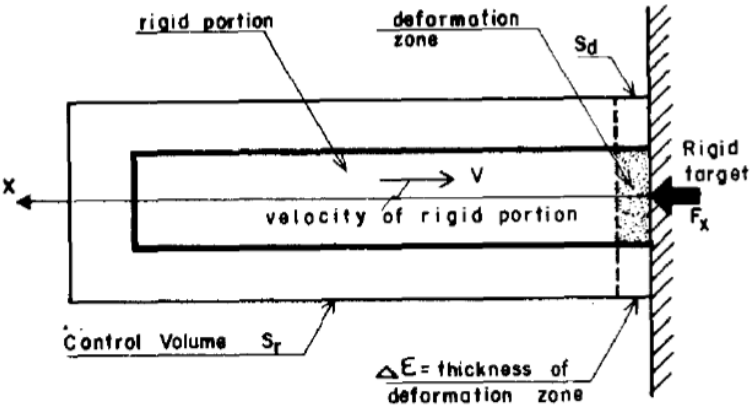


Figure 2.1: Assumed aircraft model

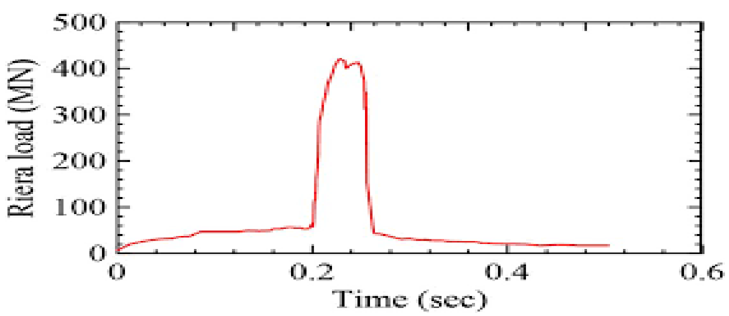


Figure 2.2: Riera reaction time plot obtained from a rigid surface

But later Abbas(1996) [2], found that using response obtained from a rigid target was highly unrealistic and miss the interaction between the structures as the effect of target yielding is disregarded. Also the Inertial and stiffness properties of aircraft are not taken into account.

H.T.Y.Yang and D.A.Godfrey(1970) [3] have found out most critical angle of impact in their study. They stated that normal strike would produce maximum reaction. They also found that it is not logical to take maximum aircraft velocity as it is difficult for large aircraft to maneuverer with high velocity near the structures with not very great height. They concluded that speed of 104 m/s is a conservative upper limit.

Kukreja (2005) [4] studied damage evaluation of a 500 MW Indian pressurised heavy water reactor nuclear confinement for aircraft (Boeing 707-320) impact and found out that thickness of containment must be at least 1.36 m to limit the damage to internal structures. While the US nuclear regulatory commission concluded that thickness of minimum 1 m would be appropriate.

Jorma and Nicolay (2007)[5] conducted numerical assessment of Boeing crash on an imaginary building using LS- DYNA software. They distributed wing load over entire length of aircraft and compared the results with Riera (1968) force history method.

Iqbal (2011) [6] predicted response of Boeing 747-320 on BWR Mark III type nuclear confinement. He observed that maximum displacement of 46 mm at the junction of dome and cylinder and 88.9 mm at mid of cylindrical portion is caused.

Iqbal (2014) [7], performed damage analysis on a nuclear confinement at multiple spots using Phantom F4, Boeing 707-320 and Airbus A320 aircrafts. Maximum damage was observed when aircraft crash at mid height of confinement. Phantom F4 behaved as a rigid missile causing punching failure if stroked with higher velocity, while Boeing 707-320 and Airbus A320 behaved as a soft crushable missile resulting in global deformation.

In all these researches associated risks such as fire, which is also a potential mean of damage due to aircraft collision is not taken into consideration.

CHPTER 3   INTODUCTION TO ABAQUS FEA SOFTWARE

Abaqus is finite element software which is capable of performing many kinds of analyses like structural, thermal, fluid and couple-field. It is a powerful tool which is widely used by Engineers in industry as well as in research. Abaqus was developed by Dassault system and is part of their SIMULIA packet. The software offers a complete simulation tool, and contains analysis tools for investigating linear problems and static events and low speed dynamic events (Abaqus standard/Implicit) or high dynamic events such as crash simulation, blast simulation (Abaqus Explicit).

For this research, Abaqus/Explicit is used, as dynamic simulations have been performed at high state. In the pre-processing phase, modelling is done using Abaqus/CAE (Computer aided Engineering). Abaqus/CAE helps user to create parts, assemblies (arrangement of parts), meshing, materials, interactions and apply loads and constraints which is automatically generated as an input file or job file. When the job or input file is ready for simulation, Abaqus/explicit takes over and uses advanced solver algorithms to solve the input file. The output file after the performed simulation is saved as odb file. Finally this file can be post-processed (visualised) in Abaqus/CAE.

3.1 Finite Element Analysis

In complex problems, it is difficult to find their exact solution. Instead, we find their approximate solutions. It can be done by two methods:

i. Weighted function method - In this we calculate approximate solution. The difference between exact and approximate solution is called error. So we select some weighted functions such that weighted integral of error over a domain is 0. It can be categorized into two:

a. Analytical approach- The solution obtained is in terms of equations or formulas.

b. Numerical approach- We get numerical values of unknowns as a solution in a domain. The methods used are-

1) Finite element Method
2) Boundary element Method

ii. Finite difference method- In this method, derivatives are expressed as differences.

Finite element analysis is a mathematical technique to solve a boundary value problem from a differential equation.

In FEM, a structure is divided into small number of elements of definite geometries. Each element consists of nodes at the corners. Also there are some integration points within the elements. Software compute variables like stresses and displacements at each integration point. Since a node can be common for multiple elements. Therefore, the variables are computed at each node by interpolating the results of the integration points near them.

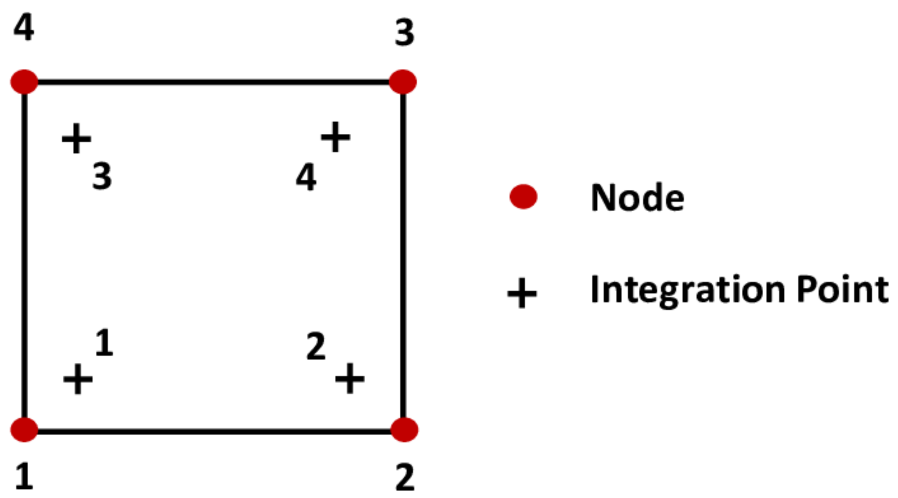


Figure 3.1: Nodes and Integration points

3.2 Types of analysis

In Abaqus, on the basis of inertial forces developed, three types of analysis are possible:

i. Static
ii. Quasi-static
iii. Dynamic

i. Static analysis- These are the analyses in which no time dependent forces are there and no inertial forces can develop.
   Equation F= [K]u is valid. They can be solved by 2 methods-
   a. Linear perturbation step- Used for buckling, steady state dynamics, frequency etc
   b. General Static step- Used for variable loading
ii. Quasi-static – In these analyses, load is applied so slowly such that structure gets deformed very slowly. So the impact of inertial force is very less and hence can be ignored. Mathematically it can be said that when
   $\omega_{excitation} < \frac{1}{3}(\omega_n)_{min}$, Quasi-static analysis must be used.
iii. Dynamic analysis - In these analyses, time dependent forces act in a short interval of time. Hence inertia has to be taken into account.
   Equation $[M]\ddot{u} + [K]u = F$ is valid. Mathematically it can be said that when
   $\omega_{excitation} > \frac{1}{3}(\omega_n)_{min}$, dynamic analysis must be used. Dynamic analyses are used to solve 2 kinds of problem: Wave propagation problem (like crash, drop, and impact) and structural dynamic problem.
   They can be solved by 2 methods- explicit or implicit.

3.3 Implicit and Explicit Analysis

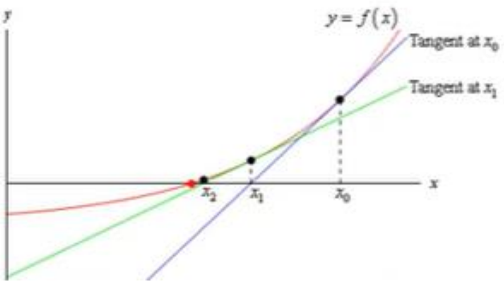
In abaqus, Dynamic analysis can be performed by two methods: Implicit and Explicit Mathematically, Implicit functions are those in which variables cannot be separated. In abaqus, implicit analysis is solved using inverse of stiffness matrix while in explicit analysis; inverse of mass matrix is computed in governing equation.

$$[M]\ddot{u} + [C]\dot{u} + [K]u = F \tag{3.1}$$

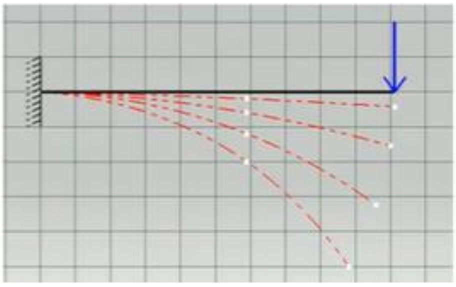
$u = [K]^{-1}\{F - [M]\ddot{u} - [C]\dot{u}\}$ (Governing equation in implicit analysis)

$\ddot{u} = [M]^{-1}\{F - [C]\dot{u} - [K]u\}$ (Governing equation in explicit analysis)

Newton Raphson method is used to solve the governing equation in implicit analysis while explicit analysis can be solved by Direct solution. In implicit analysis each time increment has to converge but it can have long time interval. In explicit analysis, it is not necessary to converge each step but their time increment must be very small.



Implicit analysis



Explicit analysis

Figure 3.2: implicit and explicit analysis

Implicit analyses are unconditionally stable while in explicit analysis, we have to check critical time step.

$$\Delta t < \Delta t_{cr}$$, where $\Delta t_{cr} = l_e/C$

$$C = \sqrt{\frac{E}{\rho}}$$

Where,

Δt is solution time step

Δt_{cr} is critical time step

l_e is characteristic length (minimum length) of element

C is wave propagation speed

This time increment Δt has to be very small, so explicit analysis takes much increments to solve the equation.

Implicit analyses are more accurate, time consuming and requires more storage space, as they are solved by iterations.

Explicit analysis is used when the strain rate or velocity is more than 10 units/s. They are used to solve extremely discontinuous events or the contact is complex. While the implicit analysis is used when the strain rate or velocity is more than 10 units/s. They are used to solve moderate contact conditions.

3.4 Elements types in Abaqus

3.4.1 Element type by family:

i. Continuum (solid or fluid)
ii. Shell elements-
iii. Beam
iv. Truss
v. Rigid elements
vi. Membrane elements
vii. Infinite elements
viii. Connector elements

3.4.2 Element type by Modelling space:

i. 3D- It is used to define 3 dimensional solid, shell, wire or point geometries.
ii. 2D Planar- It is used to define plane stress or plane strain problem in shell, wire or point geometry.
iii. Axisymmetric- It is used to generate model whose geometry and boundary conditions are symmetric about their axis.

3.4.3 Element type by DOF:

i. For stress displacement analysis, there are 3 translational DOF and 3 rotational DOF at each node for 2D elements and 3 translational DOF only for 3D elements.
ii. For heat transfer simulation, 2 thermal DOF are considered at each node for 2D elements and 3 DOF for 3D elements
iii. In coupled thermal-stress analysis, 6 displacement DOF and 2 thermal DOF are considered for 2D elements and 3 displacement DOF and 3 thermal DOF are considered for 3D elements.

3.4.4 Element types by number of nodes:

i. Linear elements- These elements have a node at each corner
ii. Quadratic elements- These elements have a node at each corner as well as at centre of edges.

To know whether an element is linear or quadratic, we check the number of nodes in it.

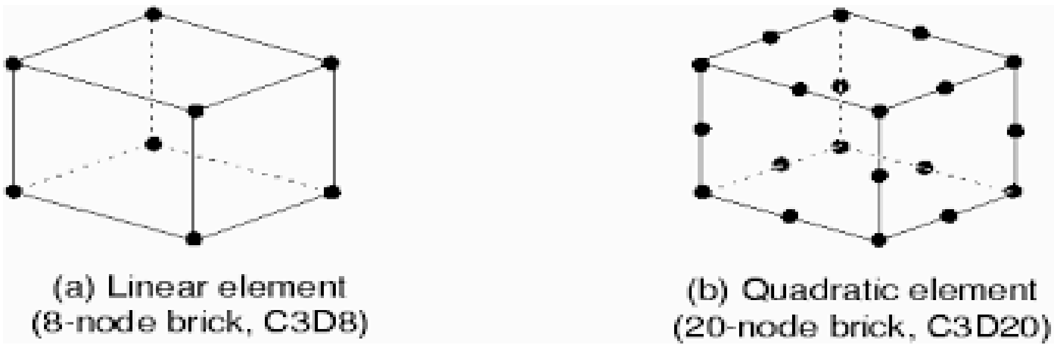


Figure 3.3: Linear and quadratic elements

3.4.5 Element type on the basis of formulation:

Mathematical theory is used to define element's behaviour. Based on that, two types of element are possible:

i. Lagrangian elements- In such elements, nodes are attached to the material. On deformation, the nodes and material undergo same displacement. They are used when distortions are small as in solids.

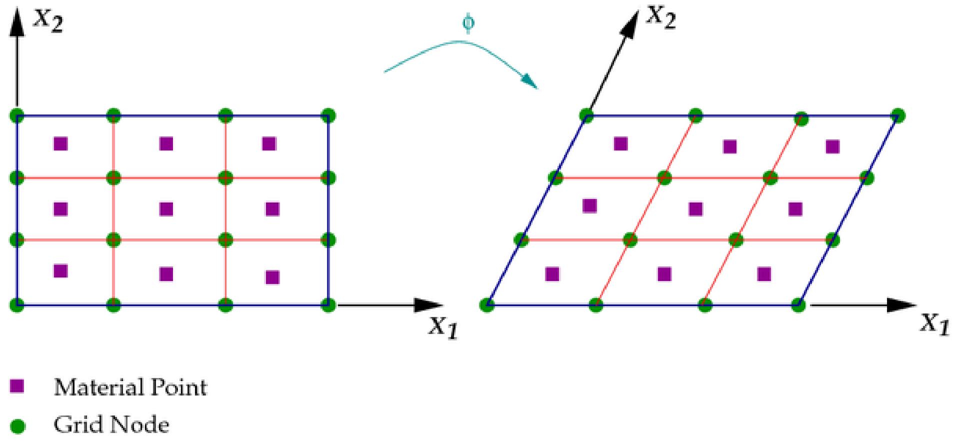


Figure 3.4: Lagrangian Elements

ii. Euler elements- In such elements, elements (mesh) remain fixed in space and the material can flow from one element to another. On deformation only material displace. They are used when distortions are small as in liquids.

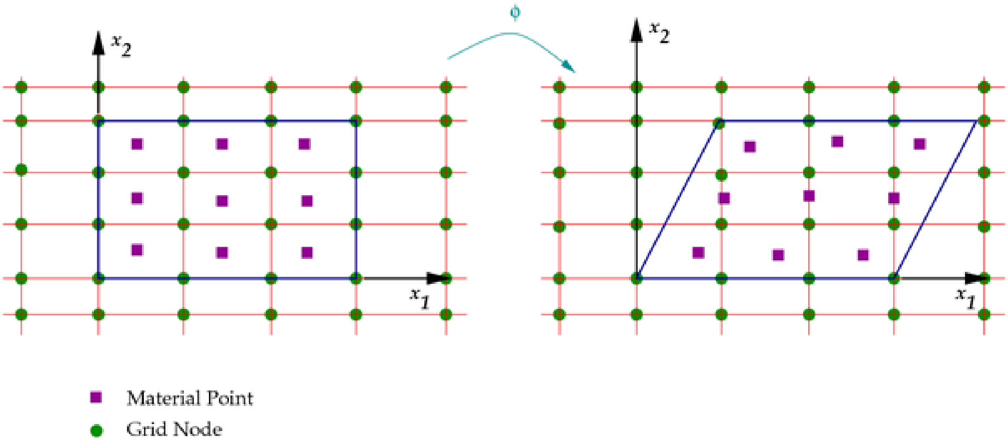


Figure 3.5: Euler element

3.4.6 Elements type on the basis of Integration point:

Integration point or Gauss point are the points in element, which are used to calculate various volumetric properties. Based on number of integration points, two kinds of elements are possible:

i. Full integration- In these elements, stiffness is calculated using all the integration point.

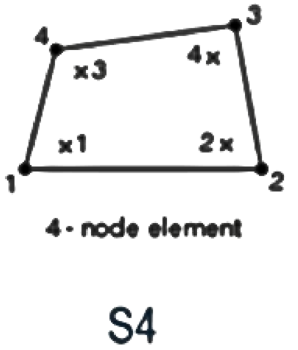


Figure 3.6: Full integration element

ii. Reduced integration – In these elements, stiffness is calculated using lesser number of integration points. In the nomenclature of these elements ‘R’ is added.

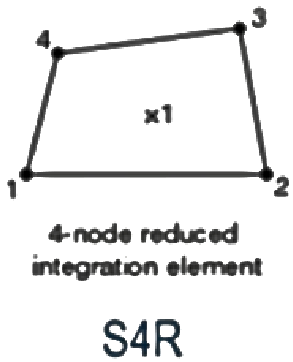


Figure 3.7: Reduced integration element

Nomenclature of elements- Elements created in Abaqus is named in following way:

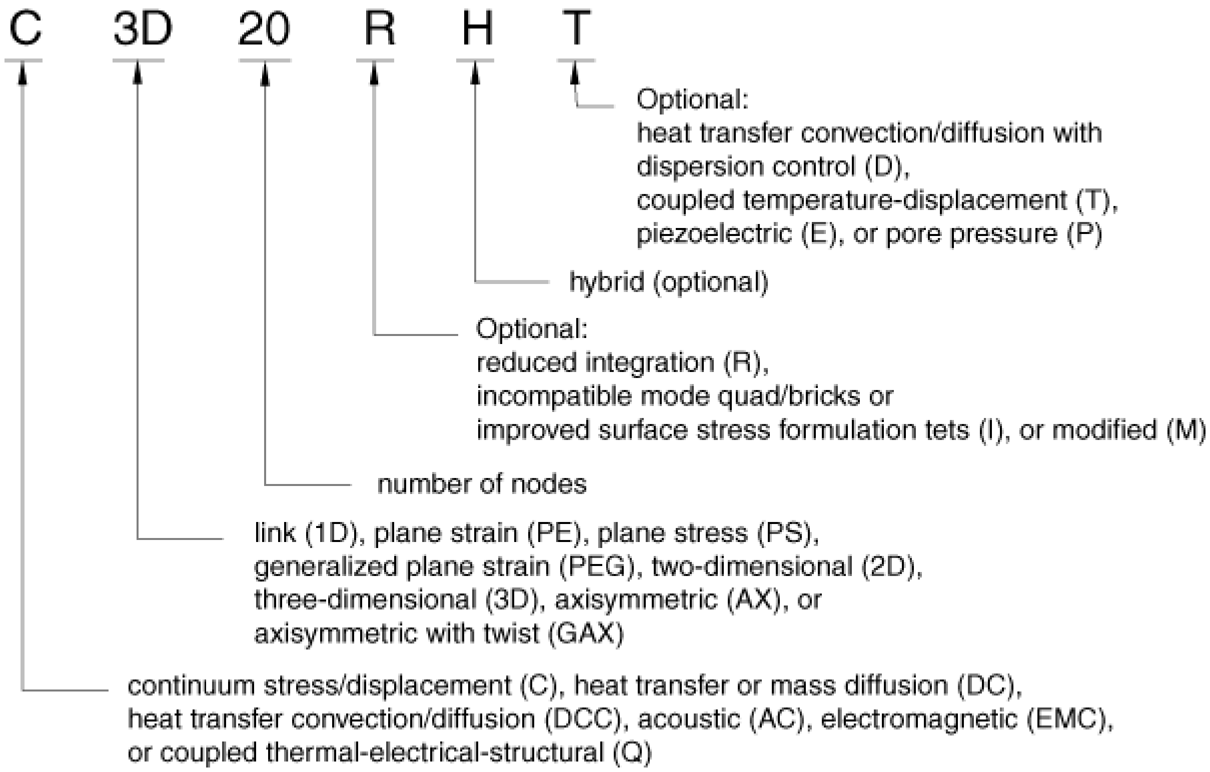


Figure 3:8 Nomenclature of elements

3.5 Meshing

Meshing is the process of discretisation or partition of a part into finite number of elements. Meshing is required in order to formulate stiffness matrix. Stiffness of regular shaped geometry can be computed easily, but in order to find stiffness of irregular geometries, member is divided into small regular shapes (elements). These stiffness values are used to find out displacement in an element for a particular force.

$$[F] = [K][X] \tag{3.2}$$

In Abaqus following types of elements are possible:

i. **Line-** It is used to assign 1D elements.
ii. **Quad and tri-** It is used to assign 2D elements.
iii. **Hex, wedge and tet-** It is used to assign 3D elements. Hex elements give the best results at minimum computational cost

Mesh size is also an important parameter which influences the accuracy of a job. Finer the mesh, more accurate is analysis but it will require more computational resource and time.

3.6 Element control

i. Distortion control- This is used to control negative element volume or the excessive volume change in elements.

ii. Element deletion- It allows elements to delete itself when they get fully damaged.

iii. Hourglass control- While using reduced integration of first order, when an element experiences bending, the central integration point does not get strained, while the position of nodes are changed. That means elements get distorted at 0 strain. This is called Hourglassing.

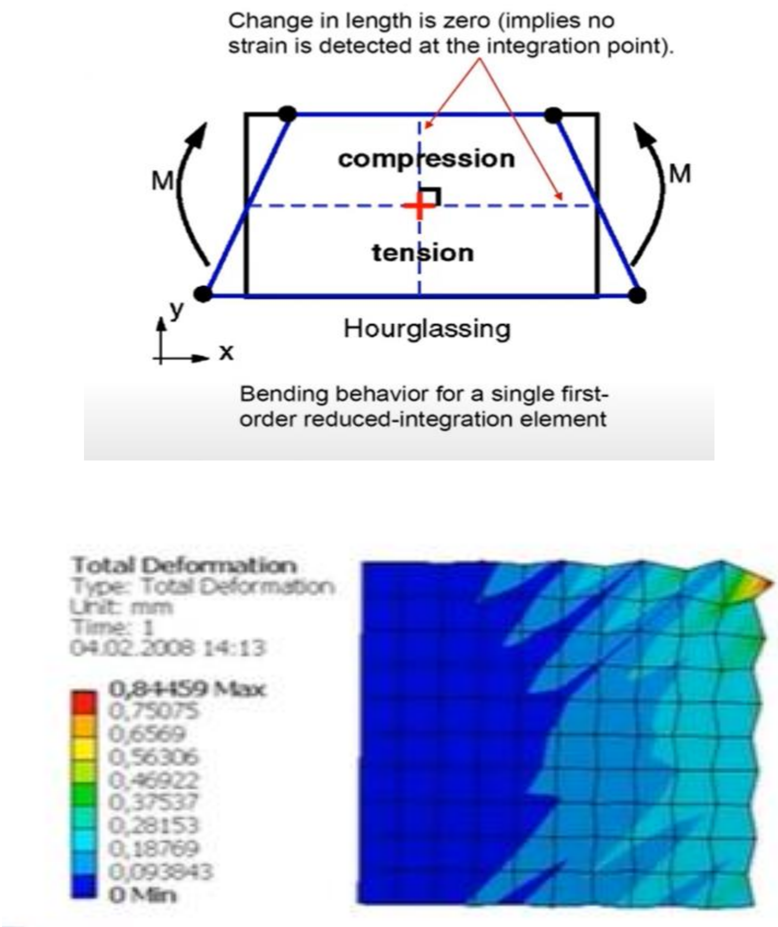


Figure 3.9: Member experiencing hourglassing

In hourglass control, 4 options are available:

a) Combined- It is used to control viscous stiffness type of hourglass control
b) Enhanced – It controls hourglassing by using enhanced strain method in solid or membrane elements with triangular or tetrahedral elements. It is used for hyper-elastic or hyper-foam elements.
c) Relaxed stiffness- It is used to control visco elastic hourglass.
d) Stiffness- It is used to control hourglassing in strictly elastic elements.
e) Viscous- It is used to control hourglassing in solid or membrane elements.

iv. Initial gap opening- This is applicable for abaqus/ Standard. Its value is kept equal to initial gap opening. For a Tangential flow over a cohesive element, initial gap is taken into account.

v. Kinematic split- It is a parameter to change the kinematic formulation of 8 nodded brick element. According to strain formulation and hourglass formulation, three options are available in it:

a) Average strain
b) Centroid
c) Orthogonal

In Abaqus/Standard, only average strain kinematic split can be used.

vi. Length ratio- This parameter is used only with distortion control parameter. It is used for controlling distortion in crushable elements.

vii. Maximum degradation – It is used to enter value at which an element is assumed to be completely damaged. It is used in the output of scalar stiffness degradation (SDEG), whose value ranges 0 to 1. User can provide maximum SDEG value other than 1 by changing value of maximum degradation.

viii. Second order accuracy- It is used for second order formulation of nodes. It is done for solid or shell elements. It is not used for linear analysis.

ix. Viscosity- This parameter is used for applying viscosity or damping coefficient value in a cohesive element or a connector.

x. Weight factor- This parameter is used for combined hourglass control to give different amount of contribution of stiffness and viscous hourglass control. Its value ranges 0 to 1.

xi. Reduced Integration- Element stiffness is created with the help of reduced number of integration point, however the mass matrix and loading is calculated using full integration point. It reduces computational cost and time. For linear and smooth nonlinear problems, reduced integration must be used.

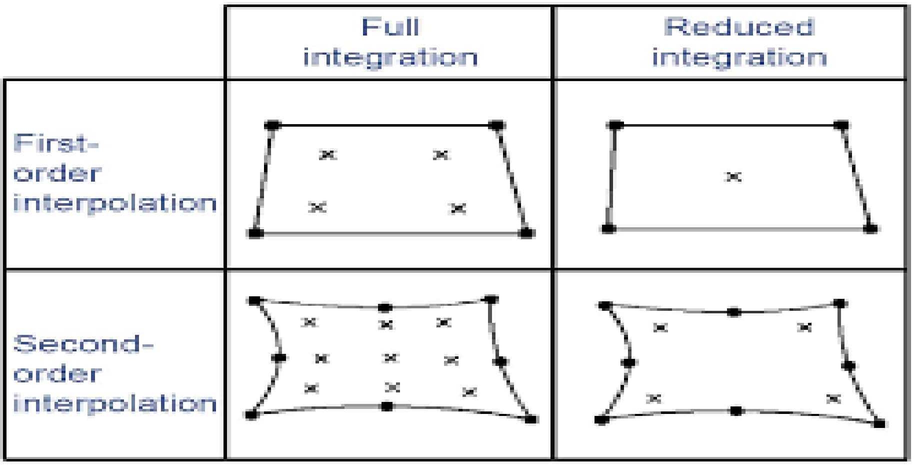


Figure 3.10: integration points

xii. Hybrid elements- These are used when a fully incompressible elements are modelled with solid elements. These elements add more variables to the problem to avoid interlocking effect. The name of hybrid elements ends with letter 'H'.

3.7 Interaction

3.7.1 <u>Contact algorithm</u>:

In order to command, how two or more instances will interact, Abaqus explicit has different kinds of contact algorithm.

i. General contact- It allows simple definition of contact with fewer restrictions on the surfaces involved. This is more user friendly as user control settings are less. We don't have to specify the master and slave surface in this algorithm, Abaqus automatically identifies them. Every surface is a potential master or slave to all other surfaces. It is less accurate.

ii. Contact pair- This algorithm is used to special kinds of problem. It is more complex than general contact algorithm. In this user specify which surfaces or nodes will come in contact with each other.

a) Node to surface contact- In this node on slave surface contact the discretised segments on the master surface. It is not possible for nodes on slave surface to penetrate on the master surface. Although nodes on master surface may penetrate on slave surface. Hence master and slave surface must be wisely chosen.

Figure 3.11: Master and slave surfaces

b) Surface to surface contact- It considers shape of both the master and slave surface in the region of contact and enforces contact condition in an average sense. Large penetration of master surface in slave surface do not occurs. Hence selection of master and slave surface is not very important.

Generally surface to surface contact provides more accurate results but involves more number of nodes for constraint. Therefore it can be computationally more expensive.

3.7.2 Mechanical constraint formulation:

When two bodies interact, a normal reaction force acts on both the bodies when they touch. If there is frictional force acting on the surface, then shear forces will develop. In Abaqus, constraint formulations are special kind of discontinuous contact, which allows force to transfer from one instance to another. They are discontinuous because they work only when two instances are in contact. There are 2 types of contact formulations:

i. Kinematic contact method- Abaqus algorithm uses this contact method by default. It is a predictor/corrector algorithm and therefore, has no influence on the stable time increment.

ii. Penalty contact method- It considers friction to be acting. Therefore it induces additional stiffness in the bodies. This stiffness may influence the stable time increment. Penalty contact method is used when either the interacting instances are analytically rigid or both the surfaces are node based.

3.7.3 Contact interaction:

Two kinds of behaviour are there at interaction-

i. Normal behaviour- It consist of:

a) Hard contact- In this formulation contact pressure is 0 at clearance when no contact has been made and it reaches infinity when contact is made. No penetration occurs in this formulation. This is most accurate formulation but numerically very challenging.

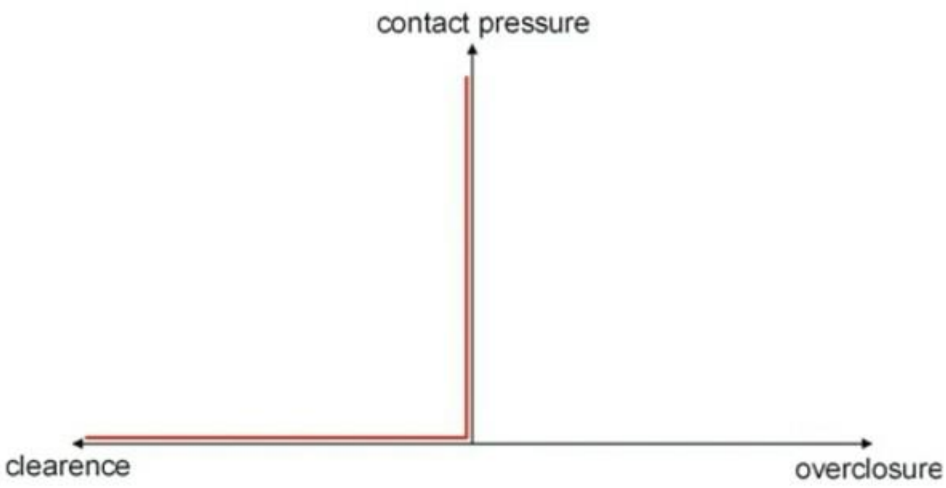


Figure 3.12: Hard contact

b) Soft contact- These formulations are relatively less challenging but are less accurate. It can be classified into 4 types depending on the contact pressure and over closure (penetration)

1) Exponential
2) Linear
3) Tabular
4) Scale factor

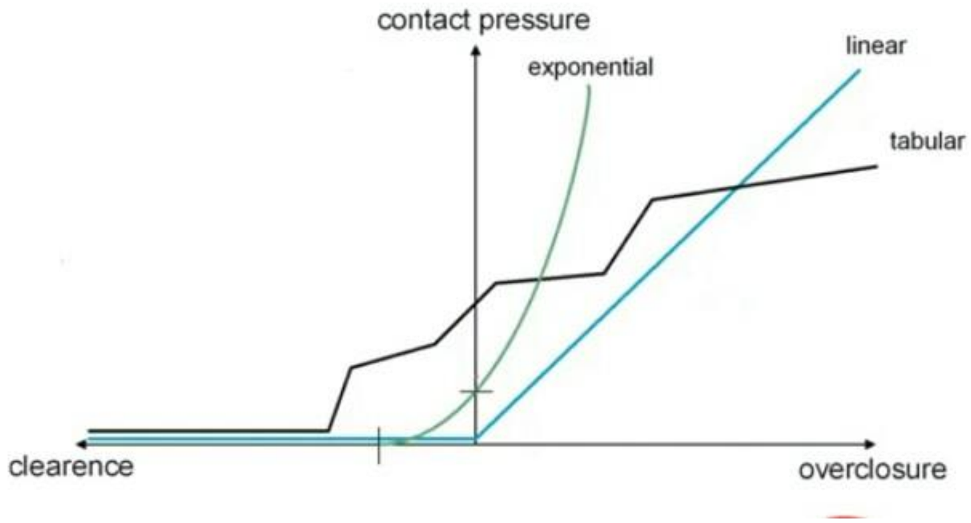


Figure 3.13: Soft contact

ii. Tangential behaviour- It consist of:

a) Frictionless- Value of friction coefficient is assumed 0 due to which free sliding of bodies take place.
b) Penalty Method- It allows surfaces to slip, even when the motion can be resisted by friction completely.
c) Static kinematic exponential decay- In this friction model, static friction coefficient decays exponentially to kinematic friction coefficient value
d) Rough- it allows infinite friction coefficient, which means there will be no slipping.
e) Lagrange multiplier- It enforces ideal friction behaviour. In this there will be no slip between the surfaces till critical value of shear stress is reached.
f) User defined- It allow user to specify shear interaction.

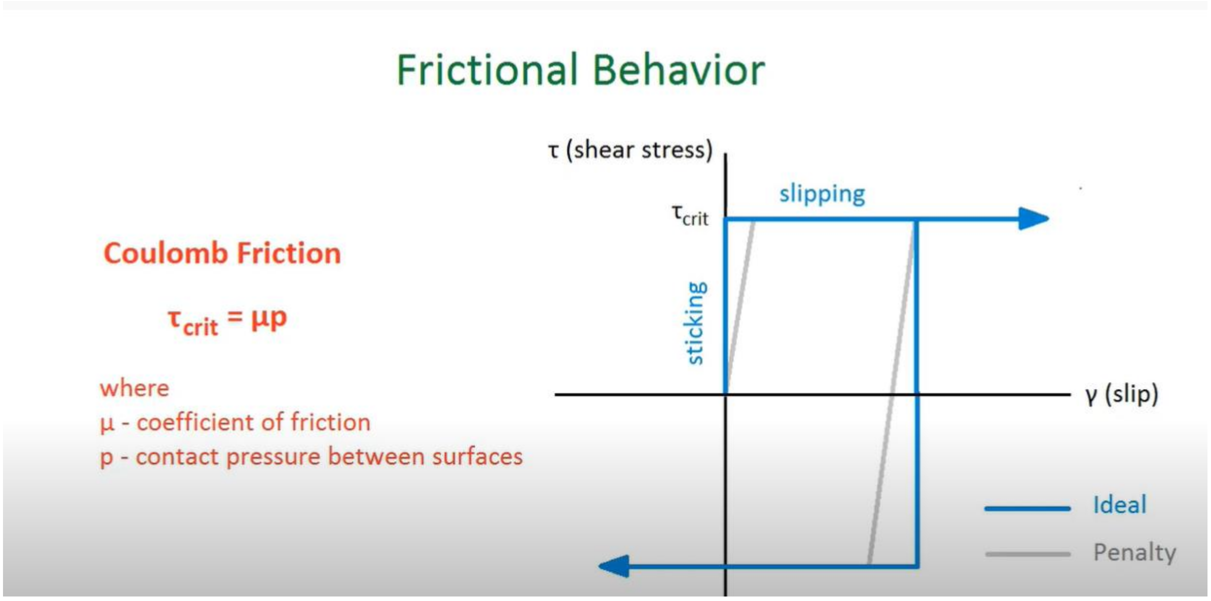


Figure 3.14: Frictional behaviour

3.7.4 Sliding:

It is used to assign relative movement between two surfaces.

i) Finite sliding- It allows for arbitrary relative separation, sliding and rotation of contact surfaces. The connectivity of currently active constraints get updated based on relative tangential motion of the contact surfaces
ii) Small sliding- It assumes that there is very small relative sliding between the surfaces. The connectivity of constraints between the nodes involved remains fixed throughout the analysis.

3.7.5 Constraints:

Constraints are the connections which partially or completely eliminate degree of freedom of the connected members.

Following kinds of constraints can be used in abaqus:

i. Tie- It ties to different surface to each other in such a way that no relative motion can occur between them.
ii. Rigid body- It allows user to provide any region of the instance with infinite stiffness.

iii. Display body- It allows user to just display any instance without taking part in analysis.
iv. Coupling constraint- It allows user to constraint movement of a surface to a movement of a single point.
v. Shell to solid- It allows user to couple the movement of a shell edge to the movement of an adjacent solid face.
vi. Embedded region- It allows user to embed any instance within another instance.
vii. Equation constraint- It describes linear constraint s between individual DOF.

CHAPTER 4 FINITE ELEMENT MODELLING

Aircraft collision is a complex phenomenon which requires sophisticated 3D Finite element numerical software to analyse it completely. It is a type of impact loading, which acts for a very short span of time; thus it cannot be analysed in implicit manner as it will need huge memory and plenty of time. Hence, we use explicit analysis as it gives quick results with reasonable accuracy. Abaqus/explicit has enough number of material models, sufficient geometric library, assembling techniques, interaction properties and explicit analysis.

In order to complete the analysis and view the results, following steps are to be followed in Abaqus/Explicit:

1) Geometric modelling: (part modules)

2) Material model: (property module)

3) Boundary Condition

4) Meshing

5) Load

6) Positioning of each instance (Assembly Module)

6) Analysis type and procedure: (step module)

7) Interaction between parts: (interaction module)

8) To run a program: (job module)

9) Post processing, results: (visualization)

4.1 Modelling

4.1.1 Building:

Beams and columns are modelled as 3D deformable with base feature wire of planar type. Section assigned to them is I section with dimension 250 mm * 400 mm.

Slabs are made as 3D deformable with base feature shell of planar type. Their thickness is assigned as 200 mm.

Walls are made as 3D deformable with base feature solid of extrusion type.

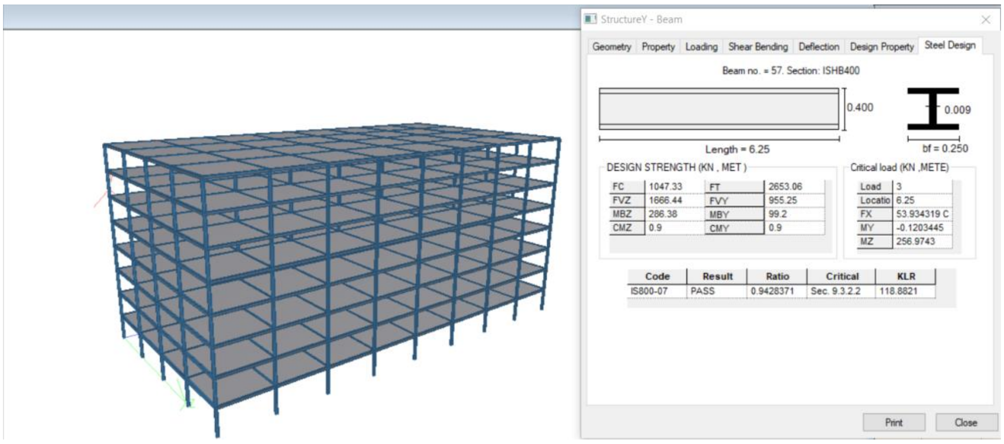


Figure 4.1: Staad building design

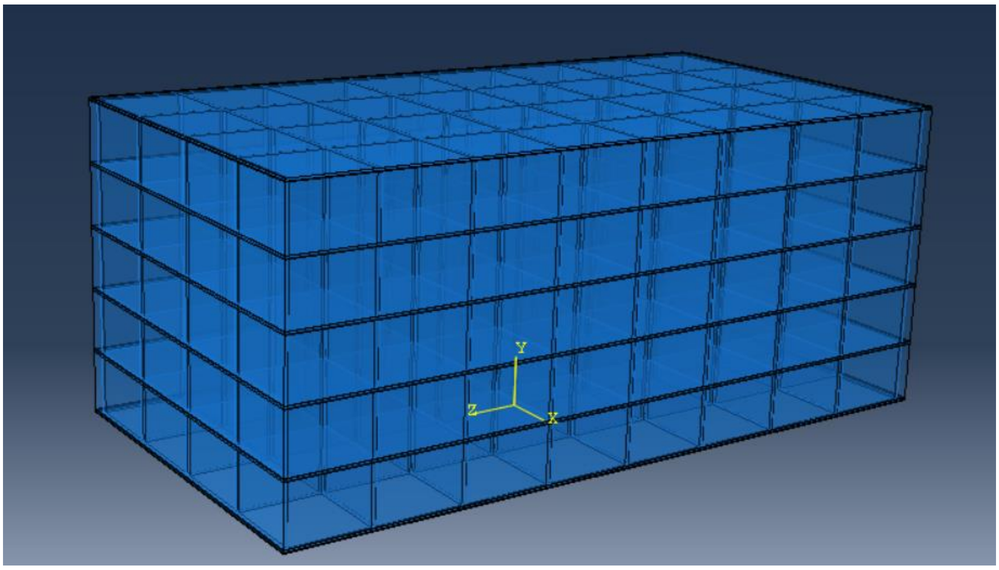


Figure 4.2: Abaqus Building model

4.1.2 Boeing 767-400 ER:

Boeing 767 has a fuselage length of 61.4 m, wingspan of 51.9 m and tail height of 19.4 m.

Aircraft Boeing 767 model has been transferred via Solidworks as IGES file. It is used as shell element of thickness 0.024 m in Abaqus.

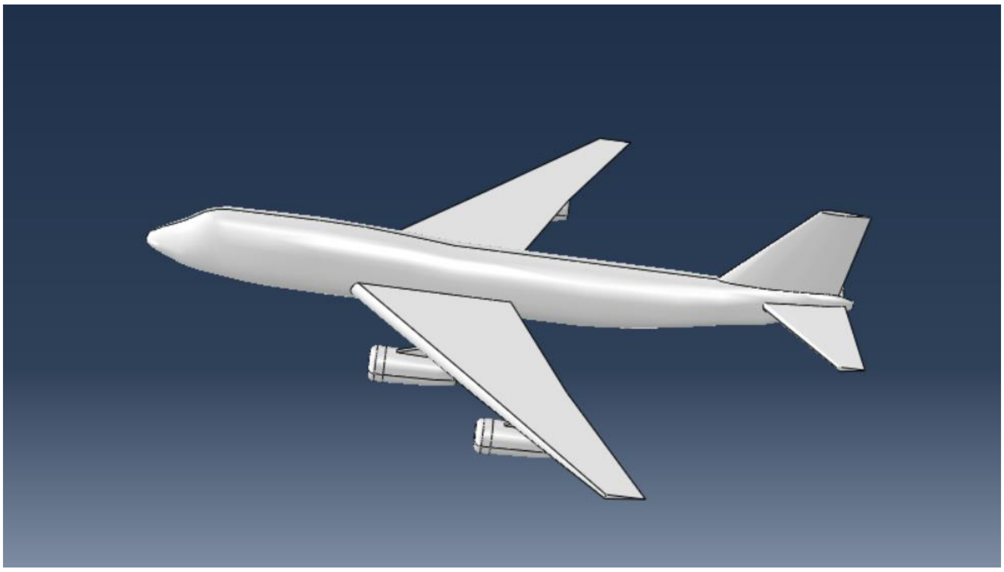


Figure 4.3: Aircraft model

4.2 Failure model

4.2.1 Concrete Damage Plasticity model:

It is an approach to model a failure of concrete or any other quasi brittle material. Quasi brittle materials are those in which failure is caused by fracture instead of large plastic flow accompanied with large fracture region. This model can be used for plain concrete or reinforced concrete structure modelling. It can be applied in case of monotonic, cyclic or dynamic loadings. It can take visco-elastic behaviour of material into account. It also allows user to control the stiffness recovery during cyclic loadings.

Through this model we can achieve a concrete failure model which is subjected to multi-axial loading.

As we know concrete is a heterogeneous material which shows a complex non-linear behaviour. In compression it exhibits ductile hardening behaviour (as there is increase in stress with increase in strain) up to a certain limit. While in tension, concrete undergoes softening (as there is decrease in stress with increase in strain). This complex behaviour must be taken into account while working on concrete models. It also allows user to control the stiffness recovery during cyclic loadings.

This model assumes that there can be two damage mechanism- tensile cracking or compressive crushing. Also the stiffness of material is reduced by damage parameters in tension and compression separately.

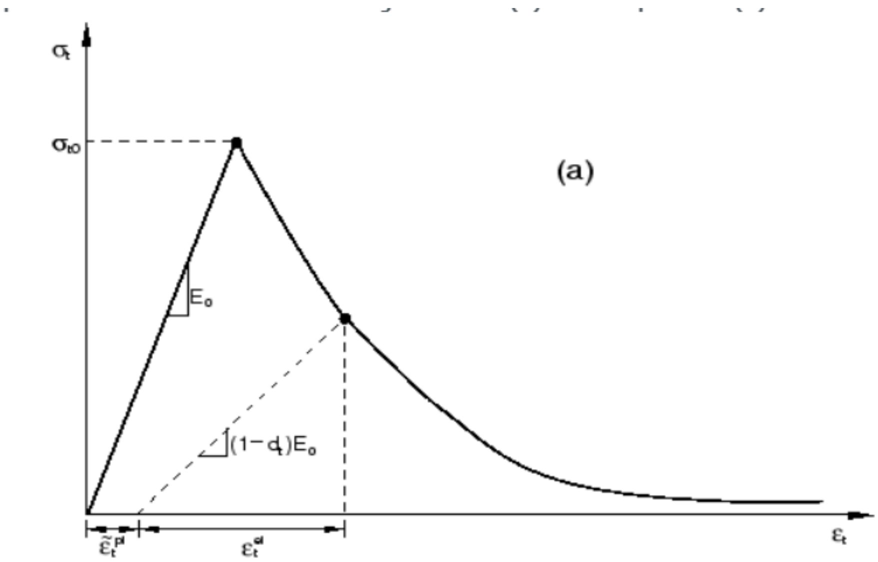


Figure 4.4: Concrete response to uniaxial loading in tension

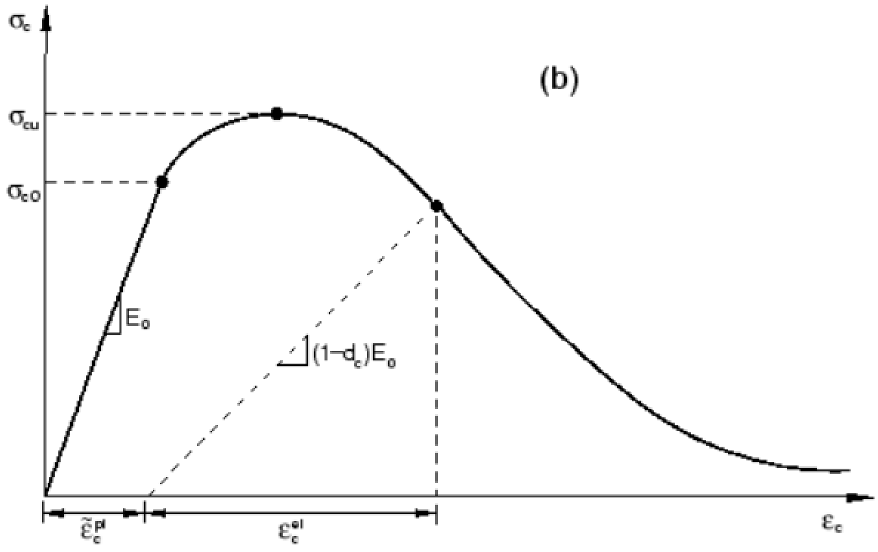


Figure 4.5: Concrete response to uniaxial loading in Compression

While using CDP model, following data are needed.

i. Tensile behaviour- Plot of tensile yield stress and cracking strain. The above curve (a) above the linear part shows this behaviour.
ii. Compressive behaviour- Plot of compressive yield stress with inelastic strain. The above curve (b) after the linear part shows this behaviour.
iii. Fracture energy- It is the energy required to open unit area of cracked surface.
iv. Dilation Angle- It is the angle of internal friction of material. It ranges between 30° to 40°
v. Eccentricity- It shows the rate at which the function approaches asymptote in the hyperbolic surface of plastic potential in meridian plane
vi. Viscosity parameter- This parameter is used to suppress the instabilities of a problem due to convergence. A lower value of viscosity parameter must be used otherwise it will result in accuracies.
vii. K parameter- It controls the shape of failure surface.
viii. σb0/σc0- It is the ratio initial biaxial compressive yield stress to initial uniaxial compressive yield stress.
ix. Damage parameter- It is a parameter which signifies the damage that has been occurred in a concrete member. If it value is 0, it means it have full reserve strength and no damage has been done. If its value is 1, it means that

the member has suffered maximum permissible damage and no strength is left. There are separate damage parameters in tension and compression.

4.2.2 Johnson Cook Model:

For steel, Johnson Cook Plasticity model was used as this is the most appropriate damage model for impact and penetration analysis. It incorporates coupled effect of strain, strain rate and temperature on flow stress.

According to Johnson Cook, equivalent stress can be expressed as

$$\sigma_{eq}=[A+B(\varepsilon_p)^n]\,.[1+C\ln(\frac{\dot{\varepsilon}_p}{\dot{\varepsilon}_0})]\,.[1-\theta^m] \qquad (4.1)$$

First term in expression controls strain.

A is yield stress under quasi- static loading,

B is the strain hardening parameter (slope of strain hardening region),

n is strain hardening coefficient (it represents curvature of strain hardening region)

ε_p is the equivalent plastic strain,

Second term in expression controls strain rate.

C is the strengthening coefficient of strain rate (it takes into account impact of change in strain rate on stress)

$\dot{\varepsilon}_p$ is the actual strain rate

$\dot{\varepsilon}_0$ is reference strain rate (experimental strain under quasi-static loading)

Third term in expression controls temperature effects.

θ is the working temperature

m is temperature coefficient.

Additional parameters D_1, D_2, D_3, D_4 and D_5 are also used to calculate Fracture strain ε_f.

4.3 Material:

It is required to define behaviour of a material that is its constitutive relationship.

4.3.1 Building- For slabs, M 35 grade concrete is used and for frame Fe 500 grade steel is used.

i. M 35 concrete

Table 4.1: Concrete properties

Density, (kg/m3)	2300
Modulus of elasticity, E (N/m2)	1.9e10
Poisson's ratio, ϑ	0.19
Dilation angle	35
Eccentricity	0.1
fb0/fc0	1.16
K	0.66
Viscosity parameter	0
Compressive strength, N/m^2	35e6
Tensile strength, N/m^2	4.2e6

a) Compressive Behaviour:

Concrete compressive behaviour is obtained from experimental study using cube/ cylinder test.

Table 4.2: Yield stress Vs Inelastic strain

Yield stress	Inelastic strain
17500000	0
20420381	9E-006
23201699	2.6E-005
25794710	5.3E-005
28147291	9.2E-005
30208536	0.000147

31933400	0.000219
33287268	0.000312
34249664	0.000425
34816428	0.000559
35000000	0.000713
34827785	0.000886
34339025	0.001076
33580805	0.00128
32603880	0.001495
31458944	0.00172
30193679	0.001951
28850760	0.002186
27466761	0.002423
26071812	0.002661
24689812	0.002898
23338987	0.003133
22032631	0.003367
20779913	0.003597
19586658	0.003824
18456056	0.004048
17389276	0.004268
16385982	0.004485
15444750	0.004699
14563403	0.00491
13739267	0.005117
12969369	0.005322
12250587	0.005524
11579755	0.005723
10953744	0.00592
10500000	0.006072
17500000	0

b) Tension Behaviour

Concrete tensile behaviour is obtained from experimental study using split cylinder test.

Table 4.3: Yield stress Vs Cracking strain

Yield stress	Cracking strain
4200000	0
2330096	0.000321
1650807	0.00058
1292702	0.000821
1069362	0.001055
915842	0.001286
803371	0.001514
717171	0.001741
648848	0.001967
593266	0.002192

c) Concrete Compression damage

Table 4.4: Compression Damage parameters

Damage Parameter	Inelastic Strain
0	0
0	9E-006
0	2.6E-005
0	5.3E-005
0	9.2E-005
0	0.000147
0	0.000219
0	0.000312
0	0.000425
0	0.000559
0	0.000713
0.00492	0.000886

0.018885	0.001076
0.040548	0.00128
0.068461	0.001495
0.101173	0.00172
0.137323	0.001951
0.175693	0.002186
0.215235	0.002423
0.255091	0.002661
0.294577	0.002898
0.333172	0.003133
0.370496	0.003367
0.406288	0.003597
0.440381	0.003824
0.472684	0.004048
0.503164	0.004268
0.531829	0.004485
0.558721	0.004699
0.583903	0.00491
0.60745	0.005117
0.629447	0.005322
0.649983	0.005524
0.66915	0.005723
0.687036	0.00592

d) Concrete Tension Damage

Table 4.5: Tension Damage parameters

Damage Parameter	Inelastic Strain
0	0
0.445215	0.000321
0.606951	0.00058

0.692214	0.000821
0.74539	0.001055
0.781942	0.001286
0.808721	0.001514
0.829245	0.001741
0.845512	0.001967
0.858746	0.002192

ii. Steel

Table 4.6: Steel properties

Young's modulus; E (N/mm^2)	2e5
Poisson ratio, ϑ	0.33
Density, (kg/m3)	7850
Yield stress, A (N/mm^2)	490
B (N/mm2)	383
N	0.45
Reference strain rate, $\dot{\varepsilon}$ (s^{-1})	5e-4
C	0.0114
m	0.94
T_{melt} (K)	1800
T_0 (K)	293
Specific heat, Cp (J/kg K)	4521
Inelastic heat fraction,	0.9
D_1	0.0705
D_2	1.732
D_3	−0.54
D_4	−0.01
D_5	0.0

4.3.2 Boeing 767- Material used in airplane has following properties.

a) Density: 2780 N/m^2

b) Elastic Property: Modulus of Elasticity- $72*10^9$ N/m^2

Poisson's Ratio- 0.3

c) Plastic Property:

Table 4.7: Yield stress Vs Plastic strain

Yield Stress	Plastic Strain
600000000	0
750000000	0.001
830000000	0.004
910000000	0.006

d) Ductile Damage:

Table 4.8: fracture strain

Fracture Strain	Strain triaxiality	Strain Rate
0.05	0	30
0.01	0.4	30

4.4 Boundary Conditions

Base of the columns of building are kept restrained from rotation and displacement with respect to all degree of freedom.

In Aircraft, velocity of -104 m/s is provided in X direction.

4.5 Inertia properties

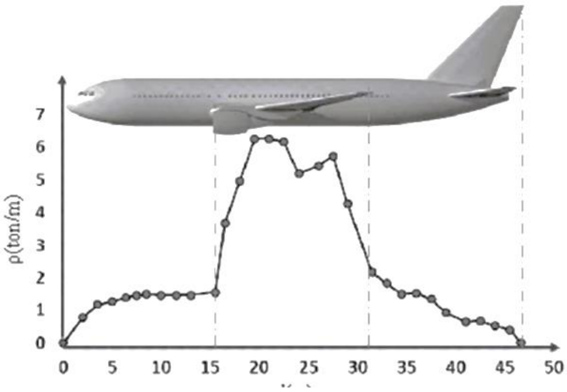


Figure 4.6: Mass distribution of Boeing 767

Mass distribution of projectile structures is one of the major factors in impact analysis therefore few inertia masses were added to aircraft at some spots such that mass distribution of the model becomes similar to actual aircraft. Mass at each node is assigned in such a way that total weight becomes 204m/s.

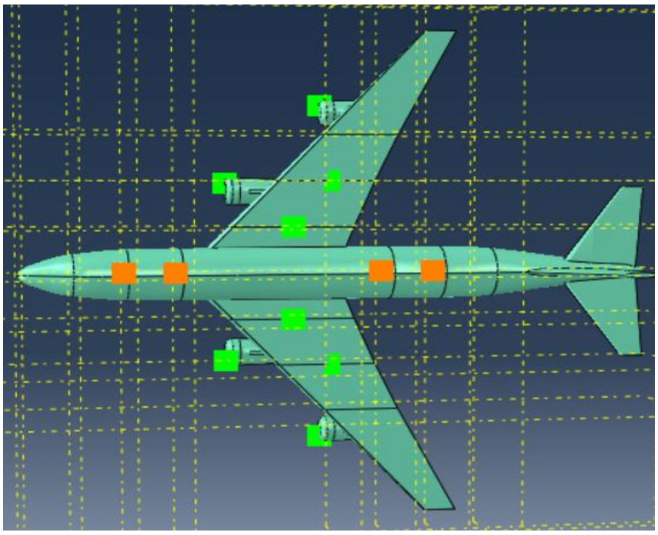


Figure 4.7: Inertia nodes added on Boeing model

4.6 Mesh

The meshing of building and aircraft is done in such a manner that we get fairly accurate results within the available computational time.

4.6.1 Building-

Since the analysis involve high strain rate, ABAQUS/ Explicit is used. Similarly for meshing, explicit element library is used. For aircraft, linear tri **S4R**: A 4-node doubly curved thin or thick shell, reduced integration, hourglass control, finite membrane strains element is used with a global mesh size of 0.45m.

For steel frame, linear **B31**: A 2-node linear beam in space is used with global mesh size of 0.67m.

For walls and slabs, linear hex **C3D8R**: An 8-node linear brick, reduced integration, hourglass control elements are used with global mesh size of 0.67m.

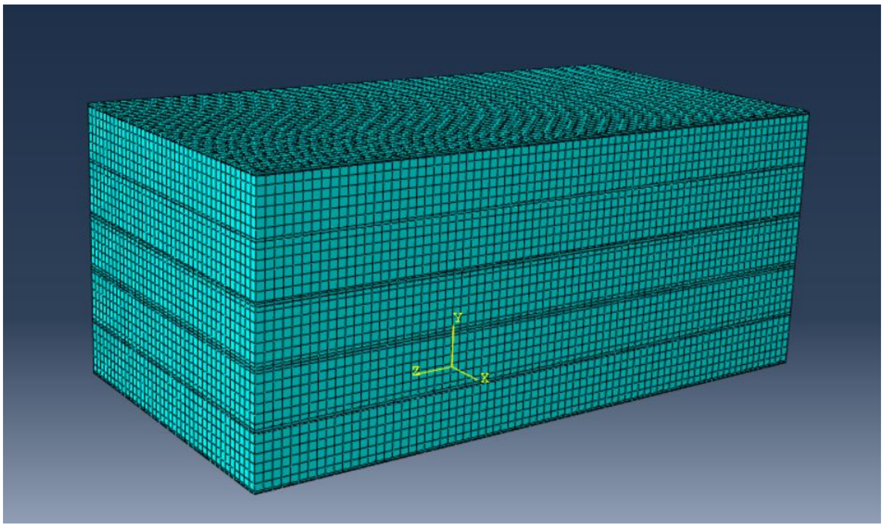


Figure 4.8: Building mesh

4.6.2 Boeing 767-

Approximate global size of mesh is chosen as 1.5m.
Maximum Deviation factor is kept as 0.1m

Element Type: C3D10M- An 10 node modified quadratic tetrahedron

Element deletion: Yes.

Element library: Explicit 3D family

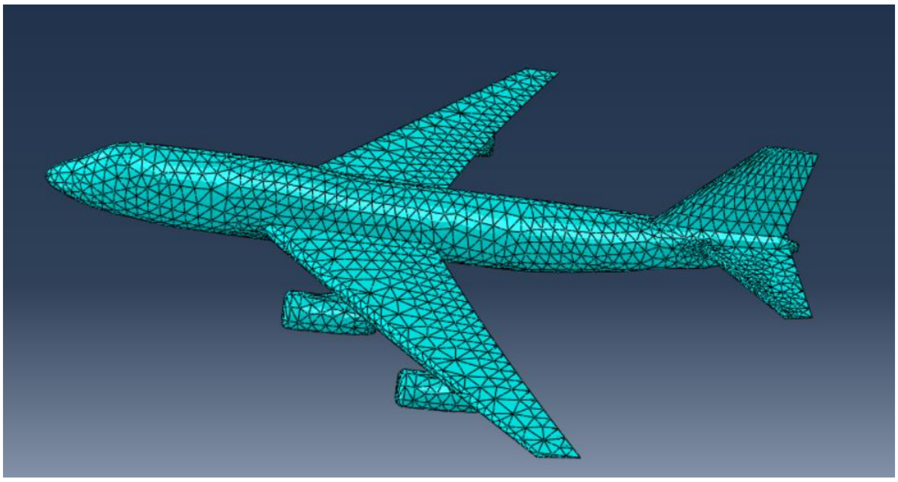


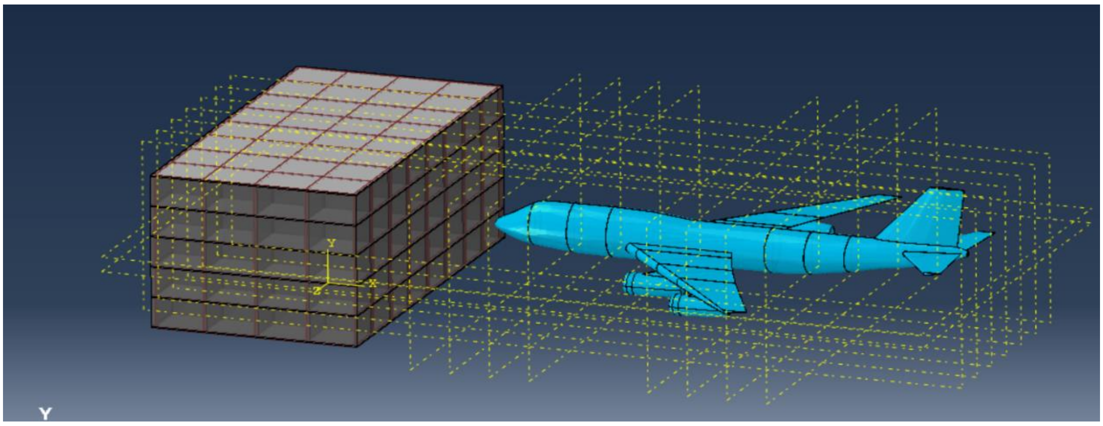
Figure 4.9: Boeing meshing

4.7 Assembly module

It is the module in which various stiffness matrices are combined to form a global stiffness matrix. In software we create instances from parts or model. Instances are the members which take part in the analysis. Instances can be of two types: Dependable and Independent

In Dependable instance, original part is used as instance. Meshing has to be performed on the part itself, which is reflected on the instance. If more than one instance is created, all will have same mesh. In Independent instance, a copy of part is used as instance. Meshing has to be performed on instance only. If more than one instance is created, each can have different mesh.

In this analysis all the instances created are dependable instance. Aircraft is positioned normal to building at distance of 3 m from face of it.



4.8 Interaction between parts

4.8.1 Interaction-

Interaction between building and aircraft is surface to surface contact (explicit).

The instance which is stiffer or which have the coarser mesh should be chosen as master surface. Here master surface is taken as building face while slave surface is selected as aircraft.

4.8.2 Contact interaction-

Normal behaviour is taken as hard contact and Tangential behaviour is taken as frictionless.

4.8.3 Constraints-

Slabs are connected to beams through tie constraint. Beams are selected as Master surface while slabs are taken as Slave surface.
Walls are connected to steel frame though embedded constraint.
Slab reinforcements are embedded in slab.

4.9 Analysis Types and Procedure

Numerical scheme followed in the analysis is explicit. In explicit scheme, we take step sizes on a very small scale in order to get stable results. This scheme is conditionally stable. Its time period is taken as 0.15 sec. In the total analysis, output is displayed in 50 time frames. Output has been requested in terms of stresses, strains, displacements and reaction terms.

CHAPTER 5 RESULTS

Numerical simulations were carried out in order to study behaviour of the steel structure, when aircraft is impact has been made at velocity of 104 m/s. Finite element analysis has been carried out using Abaqus/Explicit, wherein Boeing 767-400 ER has been considered to hit the building at three different spots.

(A) Centre of building

(B) Corner of building

(C) Top of building

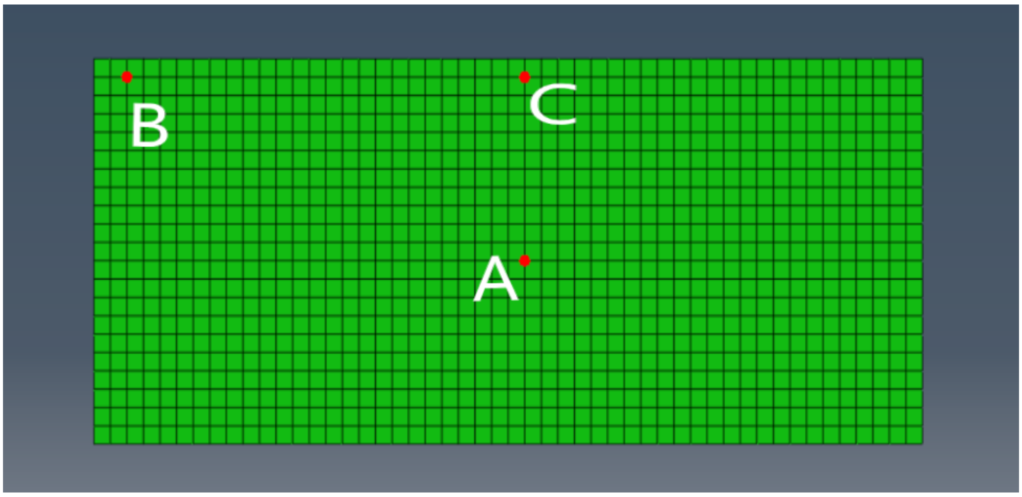


Figure 5.1: Location of strike

Various parameters like stresses, equivalent plastic strain, tensile damage and displacement are plotted with respect to time for 0.15 seconds. This time is chosen because in 0.15 s, aircraft can penetrate half the width of building if no restraint is there.

5.1 Von Miss Stress

It follows maximum distortion energy theory, which says failure occurs when maximum shear strain energy per unit volume of a structure with 3D stress state becomes more than shear strain energy per unit volume of under uniaxial loading when stress is f_y.

$$\frac{1}{12G}\,[(f_1-f_2)^2 + (f_2-f_3)^2 + (f_3-f_1)^2] \leq \frac{1}{12G}\,[(f_y-0)^2 + (0) + (0+f_y)^2] \tag{5.1}$$

$$[(f_1-f_2)^2 + (f_2-f_3)^2 + (f_3-f_1)^2] \leq 2f_y^2$$

$$f_y \geq [\tfrac{1}{2}\{(f_1-f_2)^2 + (f_2-f_3)^2 + (f_3-f_1)^2\}]^{0.5}$$

This stress $\boldsymbol{[\tfrac{1}{2}\{(f_1-f_2)^2 + (f_2-f_3)^2 + (f_3-f_1)^2\}]^{0.5}}$ is called Von Miss stress.

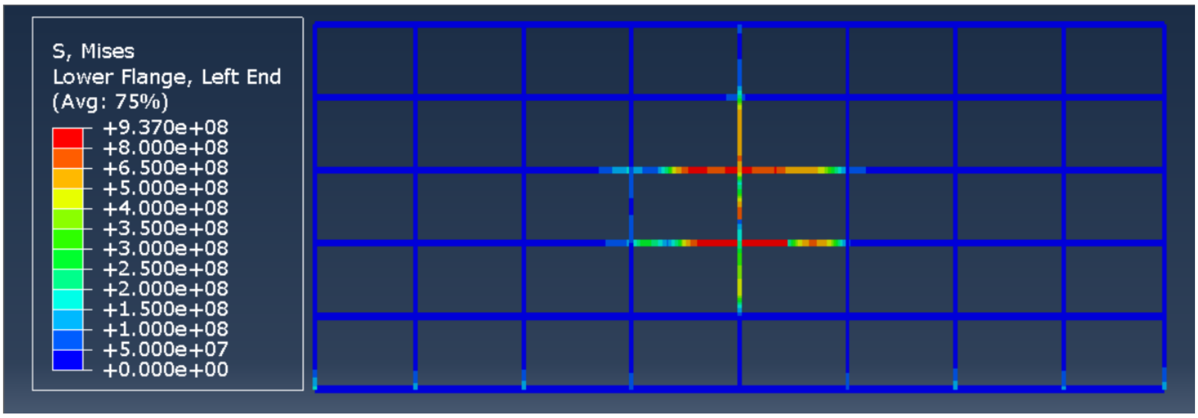
In this research, yield stress f_y is 500 MPa.

This theory is considered to be most appropriate for ductile materials. Hence it can be used in this analysis for steel framed building

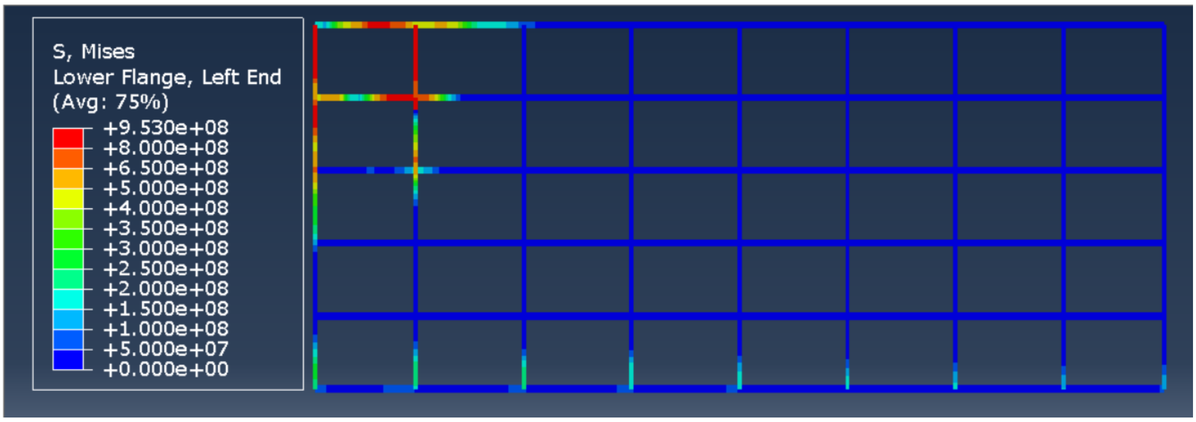
Maximum stress is developed in case when aircraft strike at corner of building (C) which is 952 MPa.

In all three cases von miss stress exceeds yield stress which is 500 MPa. Therefore it can be assumed that building will fail in all the cases.

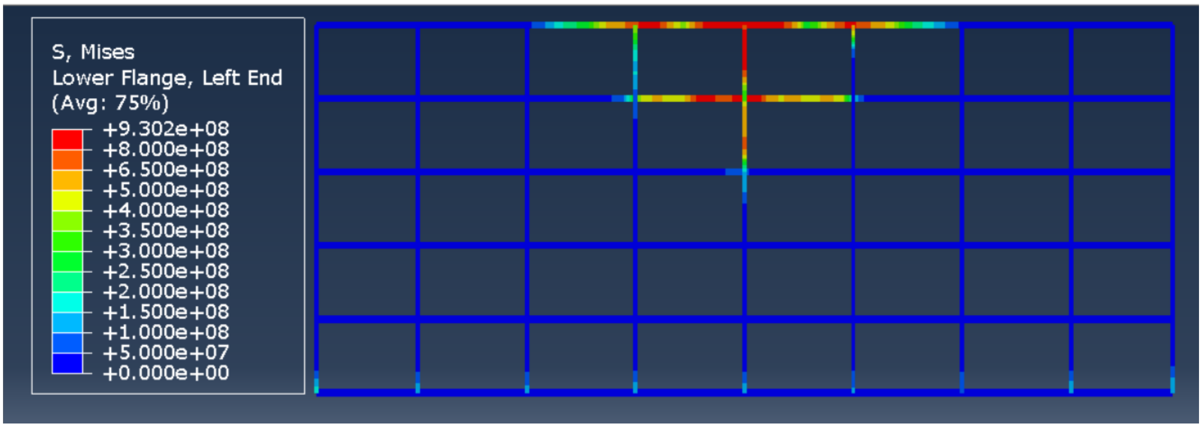
The stress contours were plotted which shows the von miss stresses in steel frame. The beams and columns which have orange and red regions have failed as the von miss stress exceeds yield stress (500MPa).



Stress contours in case (A)



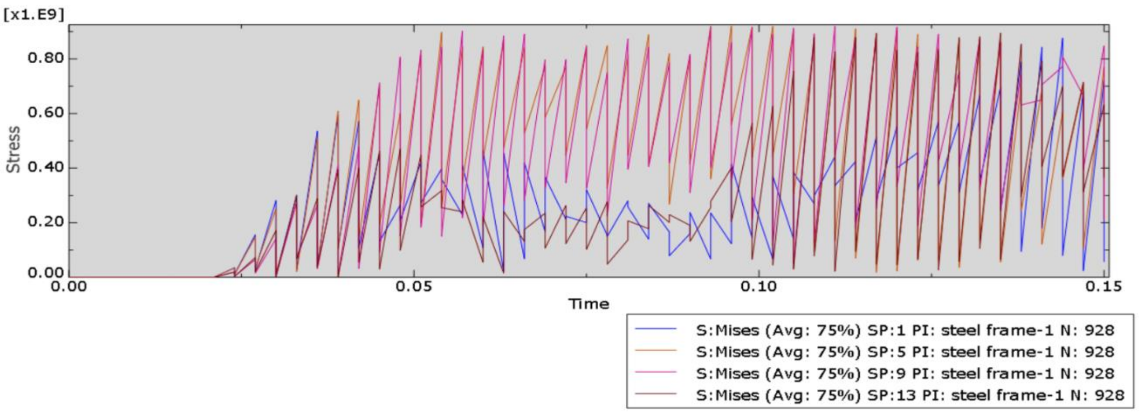
Stress contours in case (B)



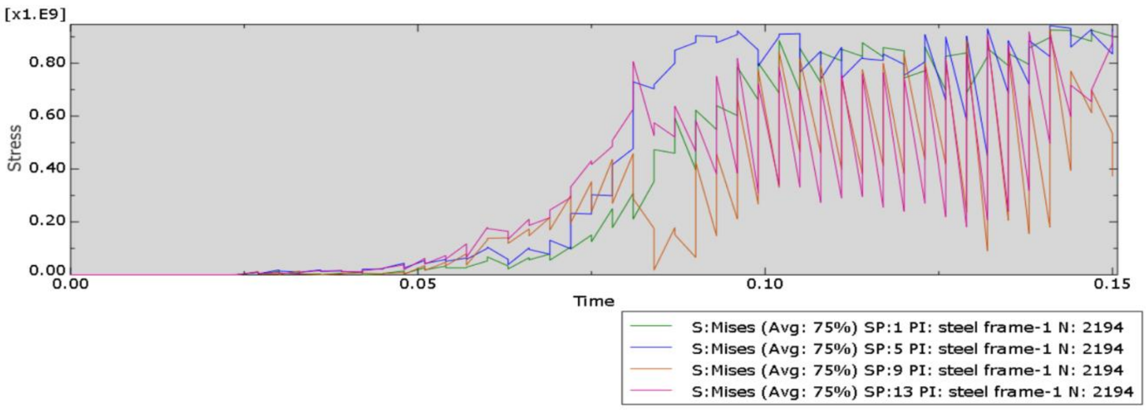
Stress contours in case (C)

Figure 5.2: Stress Contours

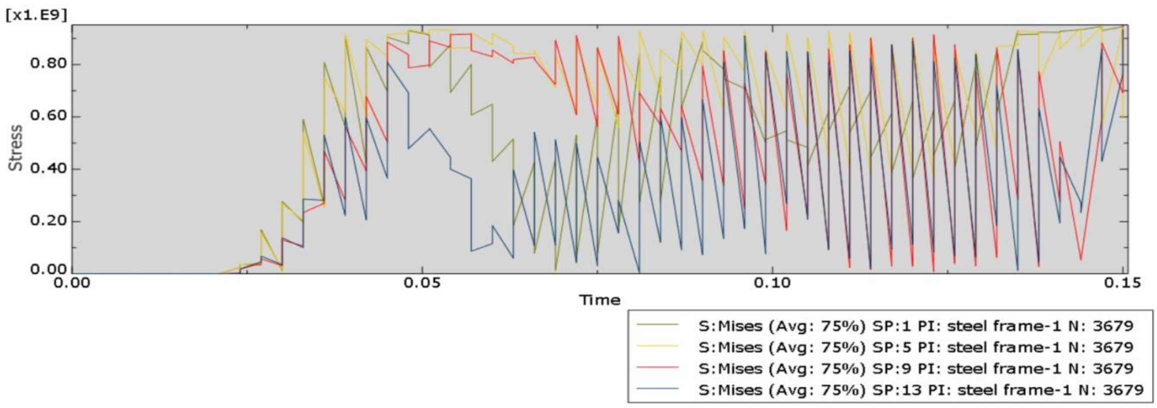
In order to examine the stress history during the whole impact period of strike, von miss stress variations at nodes in building which are directly in front of the nose of the aircraft is obtained.



Stress variation in case (A)



Stress variation in case (B)



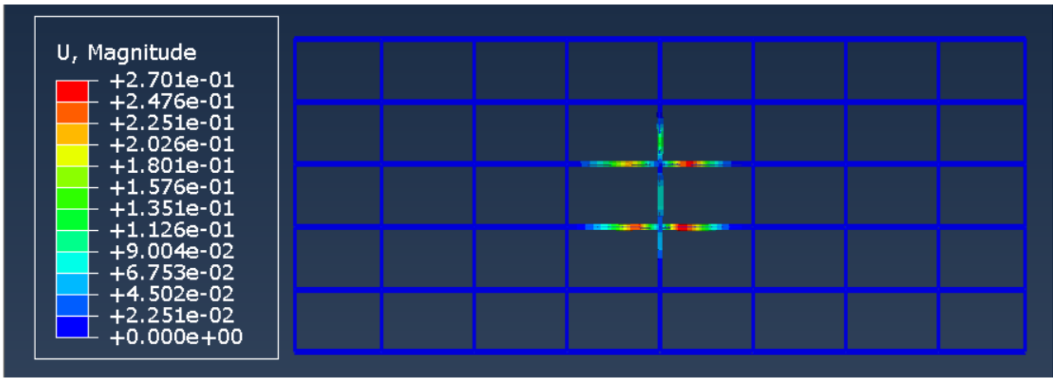
Stress variation in case (C)

Figure 5.3: Stress variation with time

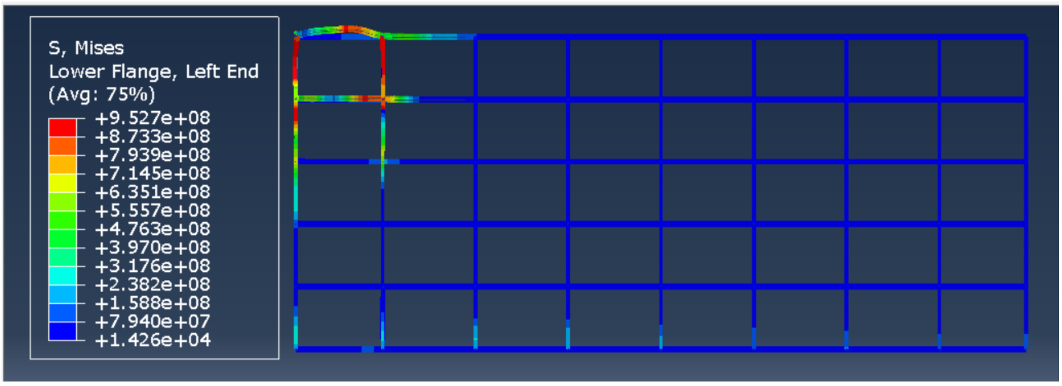
5.2 Displacement in direction of strike

Maximum displacement of each point with respect to ground is computed in each case.

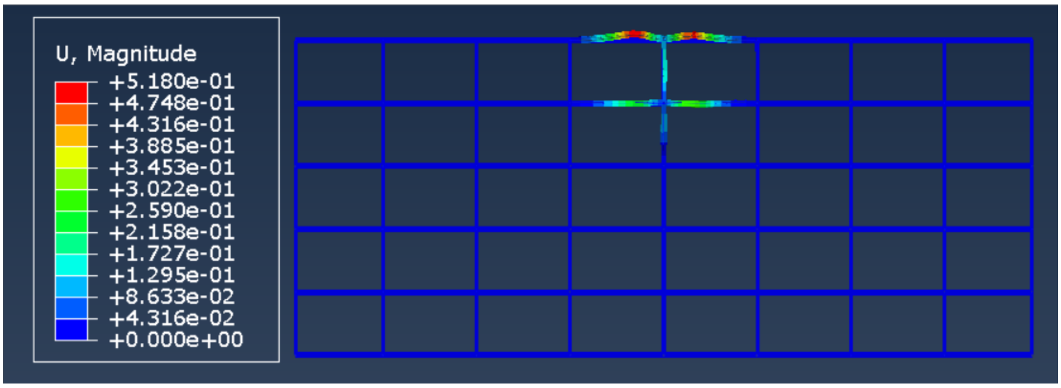
Highest displacement was found in the case when aircraft strike at corner of the building that was 64cm.



Displacement in case (A)



Displacement in case (B)



Displacement in case (C)

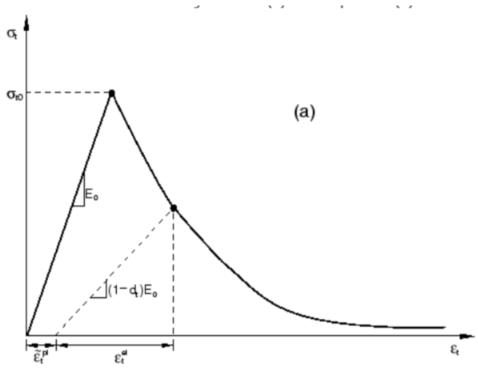
Figure 5.4: Displacement contours

5.3 DAMAGET

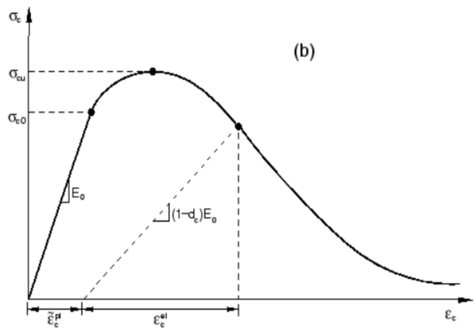
It represents tensile damage in concrete. It is a parameter which signifies the damage that has been occurred in a concrete member. If it value is 0, it means it have full reserve strength and no damage has been done. If its value is 1, it means that the member has suffered maximum permissible damage and no strength is left.

There are separate damage parameters in tension and compression. Since concrete is weak in tension, therefore only tension damage is of main concern.

DAMAGE parameters are applicable on concrete damage plasticity models only.



Tensile behaviour



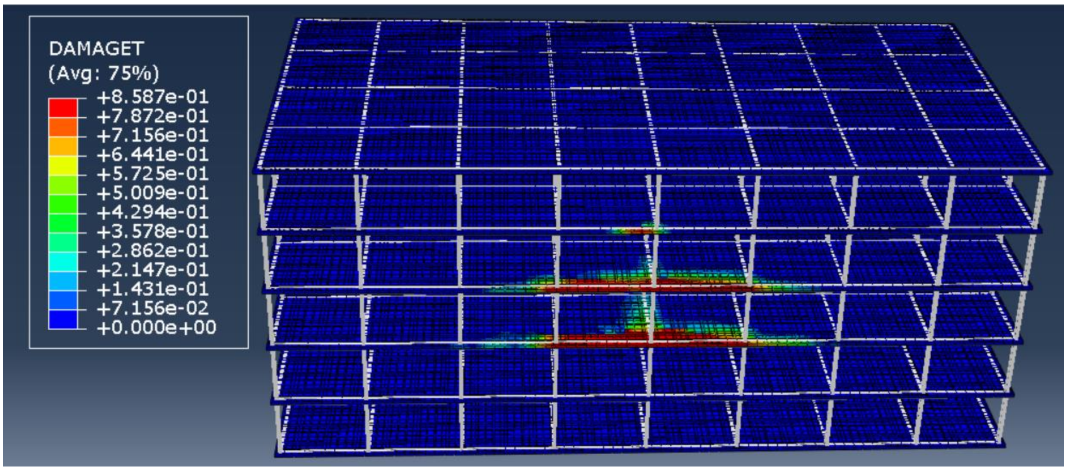
Compressive behaviour

When a concrete specimen is unloaded from any point on strain softening, its response is weakened and the stiffness seems to be damaged or degraded, which can be represented by damage parameters d_c and d_t.

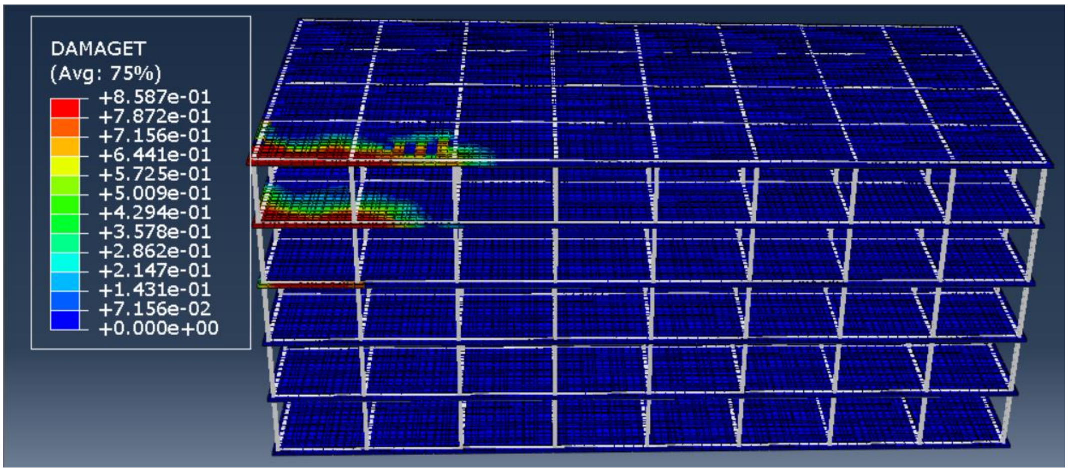
Damage parameters are functions of plastic strains, field variables and temperature. Damage parameters can be calculated as $d = (1- \frac{E}{E_0})$

where E is instantaneous elastic modulus and E_0 is initial elastic modulus.

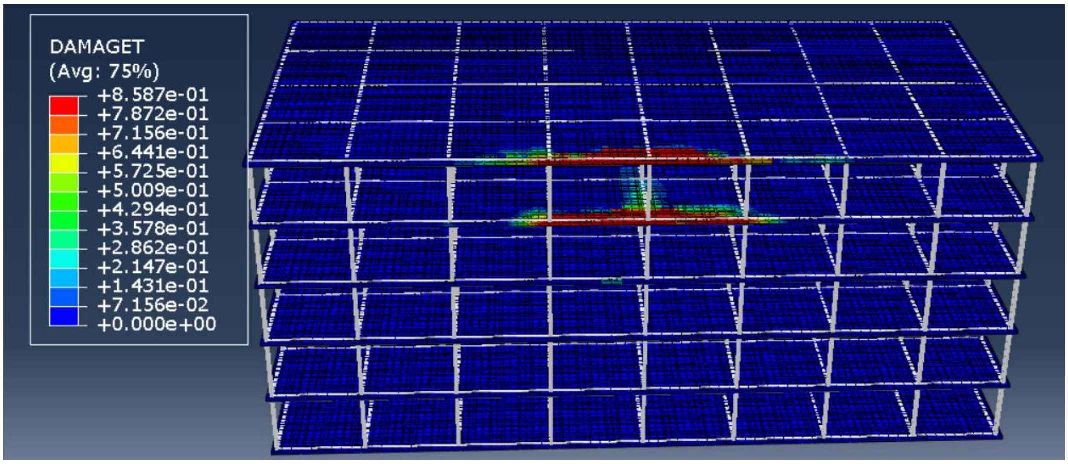
In concrete properties, maximum value of tension damage was assigned as 0.8587 by the provider. So, in the below figures, the areas in slab with red portion have damaged or about to damage in tension.



Tension damage in case (A)



Tension damage in case (B)



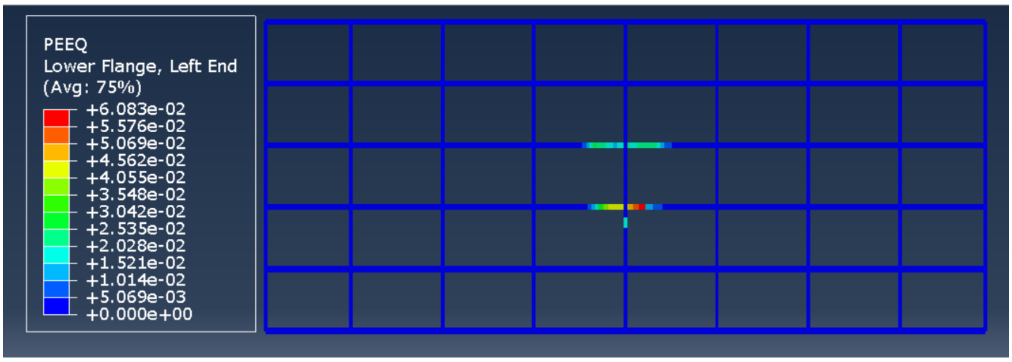
Tension damage in case (C)

Figure 5.5: DAMAGET contours

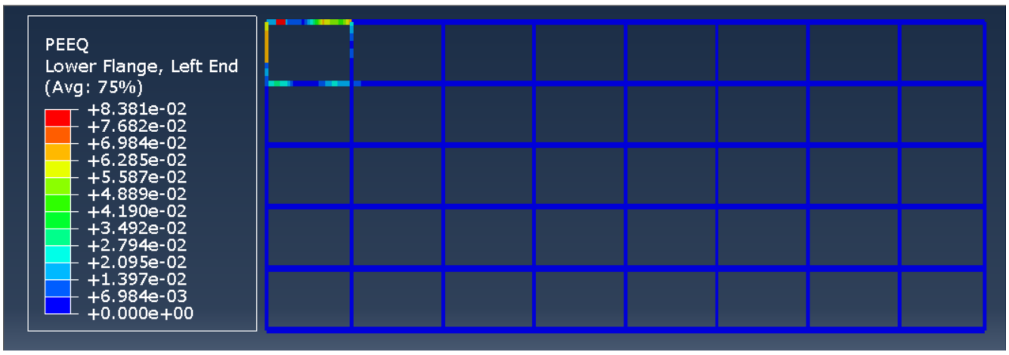
.

5.4 PEEQ

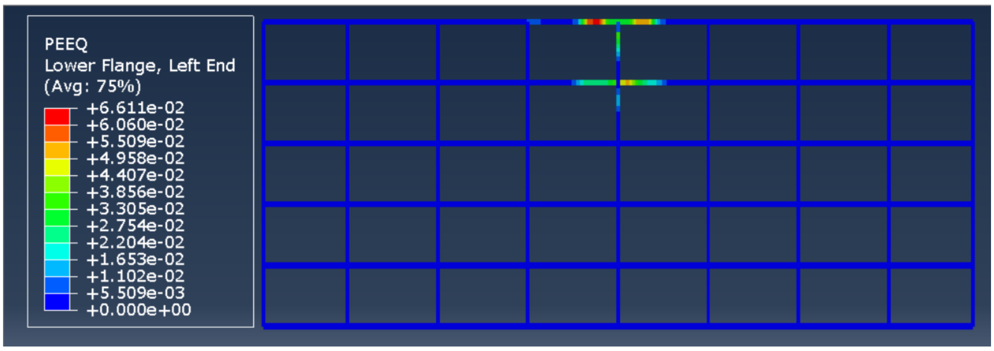
It represents plastic equivalent strain that is the strain in excess of elastic strain. It is the scalar measure of all components of equivalent plastic strains. PEEQ value greater than 0 represents material has yielded. In case of reversal loading, PEEQ adds up and continues to increase.



PEEQ in case (A)



PEEQ in case (B)

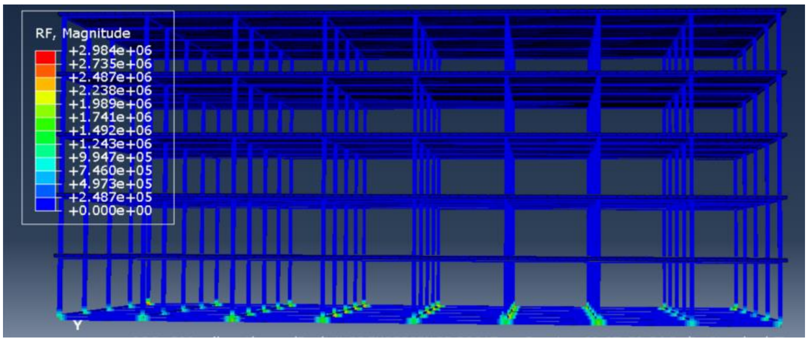


PEEQ in case (C)

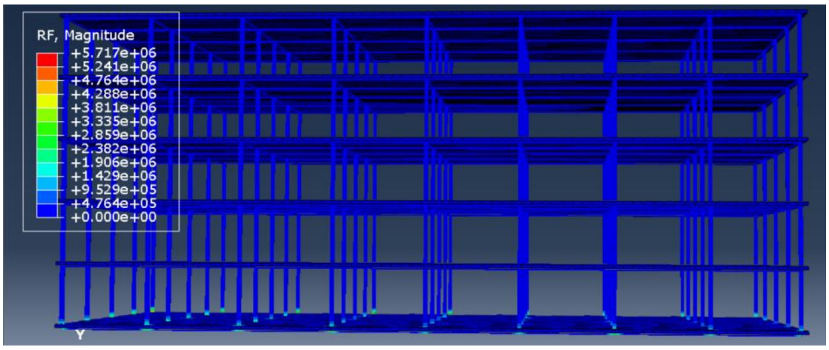
Figure 5.6: PEEQ contours

5.5 Reactions at support

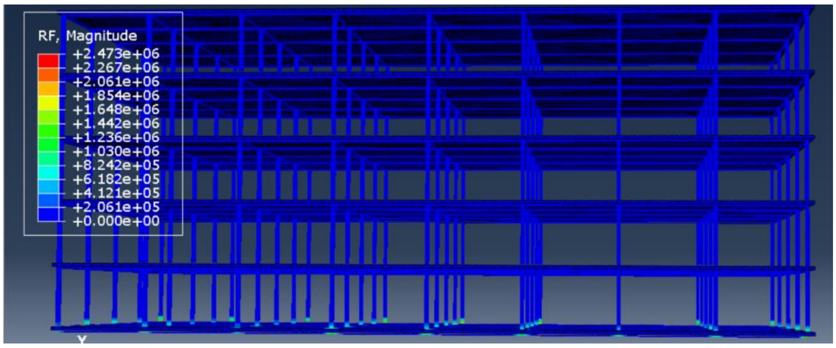
This signifies maximum magnitude of support reactions developed in the structure during any instance of analysis.



Reaction in case (A)



Reaction in case (B)

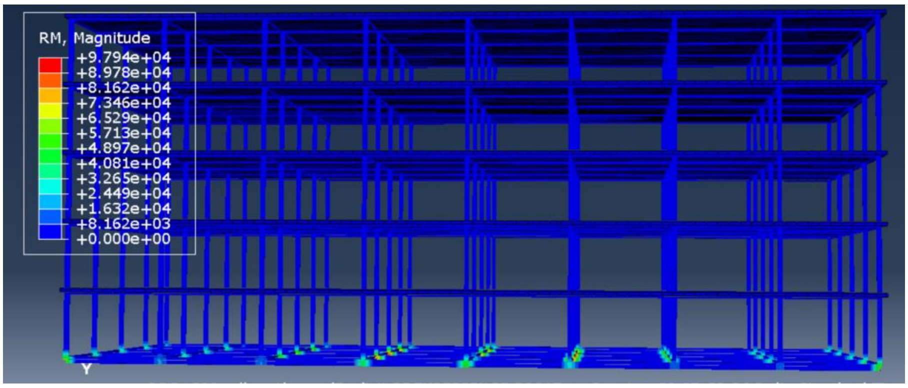


Reaction in case (C)

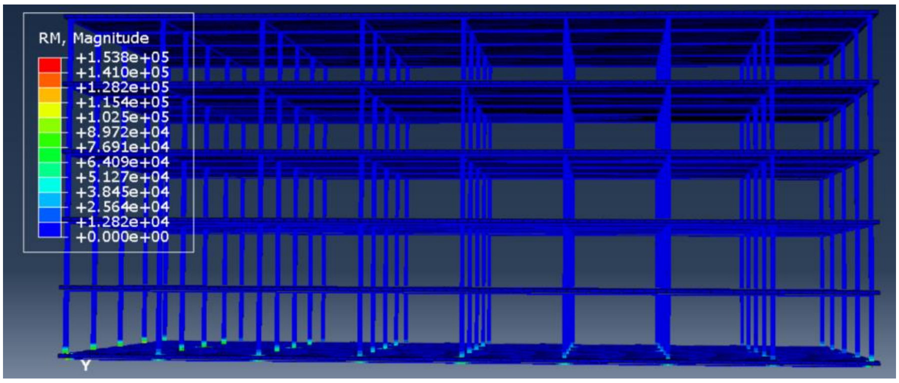
Figure 5.7: Reactions

It was found that support reactions were maximum in the case when aircraft strike at corner which is about 571 KN.

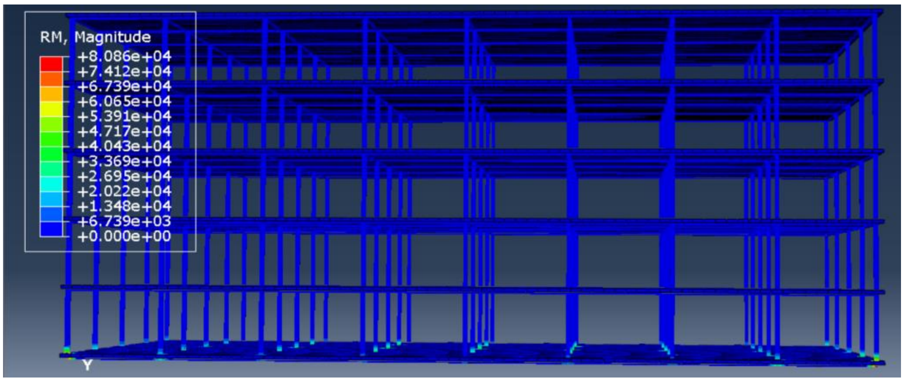
5.6 Reaction Moments at support



Reaction in case (A)



Reaction in case (B)



Reaction in case (C)

Figure 5.6: reaction moments

Maximum moment was found to be developed in case of corner strike which was 153 KNm.

CHAPTER 6 CONCLUSION

In this thesis, numerical simulations were carried out to predict the response of a rectangular steel building under normal strike of Boeing 767- 400 ER using ABAQUS/ Explicit software. Strike was made at three positions on building- Centre, top and corner.

It was found that building fails in all the three cases as the von miss stress exceeds the yield stress (500 MPa), but the stresses were highly localised. Yielded parts of building can be very easily located by examining PEEQ contours.

Corner strike was found to be most critical. Maximum von miss stress was reported to be 950 MPa and maximum deformation was 64.6 cm. Also plastic strain was found to be maximum there, which was 0.0838.

By examining the DAMAGET contours, it can be seen that some parts of slab would be damaged in tension at the immediate location of strike.

Future Prospects:

In future, damage assessment of aircraft impact on the same building when vertical bracings are added, can be done. Associated risks such as fire, which is usually followed by these kinds of events, would also be incorporated in further studies.

REFERENCES

[1] Riera, J.D., 1968, “On the stress analysis of the structures subjected to aircraft impact forces”.

[2] Abbas, H., Paul, D.K., Godbole, P.N., Nayak, G.C., 1996, “Aircraft crash upon outer containment of nuclear power plant”.

[3] Yang, H.T.Y., Godfrey, D.A., 1970, “Structural analysis of aircraft impact on a nuclear containment vessel and associated structures.”

[4] Kukreja Mukesh, 2005, “Damage evaluation of 500 MWe Indian Pressurized Heavy Water Reactor nuclear containment for aircraft impact”

[5] Jorma, A., Nikolay, D., 2007, “Analysis of aircraft impact to concrete structures”.

[6] M.A. Iqbal, S. Rai, M.R. Sadique, P. Bhargava, “Numerical simulation of aircraft crash on nuclear containment structure”

[7] Mohd. Ashraf Iqbal, Md. Redan Sadique, Pradeep Bhargava, “Damage assessment of nuclear containment against aircraft crash”.

[8] Abbas, H., (Ph.D. thesis) 1992. “Dynamic Response of Structures Subjected to Missile Impact”, University of Roorkee, India.

[9] Abbas, H., Paul, D.K., Godbole, P.N., Nayak, G.C., 1995. “Reaction-time response of aircraft crash”.

[10] Tomasz Jankowiak, Tomasz and Lodygowski, Poznan University of Technology, Institute of Structural Engineering (ISE) ul.Piotrowo 5, 60-975 Poznan, Poland, “Identification of parameters of concrete damage Plasticity Constitutive Model”.

[11] Milad Hafezolghorani, PhD Candidate; Farzad Hejazi, Senior Lecturer; Ramin Vaghei, PhD; Mohd Saleh Bin Jaafar, Prof.; Keyhan Karimzade, Msc, Department of

Civil Engineering, Faculty of Engineering, University Putra Malaysia, Selangor, Malaysia, “Simplified Damage Plasticity Model for Concrete”.

[12] ABAQUS Explicit user manuals Version 6.8

[13] Nucl. Eng. Des. 160, 13–50. Boeing Commercial Airplanes.

[14] https://modernairliners.com/boeing-767/