# Study of Behavior of RCC Beam under Impact Loading and Effect of Hourglass Energy by Finite Element Analysis using ANSYS

M Tajamul Islam[1]
[1]Research scholar,
Department of Civil Engineering,
DBIT, Dehradun Uttarakhand

Prof. (Dr.) Omprakash Netula[2]
[2]Professor and Director
DBIT, Dehradun Uttarakhand

Er. Ashuvendra Singh[3]
[3]Associate Professor and Head,
Department of Civil Engineering,
DBIT, Dehradun Uttarakhand

***Abstract:-*** **This paper discusses about the behavior of RCC beam in explicit dynamics which is a high speed analysis solver in ANSYS. Basically it is not a slow compression test, rather an impact on the concrete beam with reinforcement. The analysis is carried in ANSYS which employs Finite Element Method for analyzing the beam. Finite Element Method finds its application in the multiple areas of Engineering and Natural Sciences and beyond. Apart from having a sound mathematical foundation, the Finite Element Method has contemporary developments that span the spectrum for an extensive range of problems ranging from construction and analysis of stable convergent methods to those directed at specific applications. The aim of this research paper is to provide a brief idea of Finite Element Method while analyzing the RCC beam by using an explicit dynamics approach with the effect of hourglass energy in ANSYS along with the emphasis on basic formulations, their analysis, properties and numerous solicitations.**

***Keywords: RCC beam, Hourglass energy, ANSYS, FEA, Explicit Dynamics, Nodal Velocities***

## I. INTRODUCTION:

Beams are primarily the flexural members and carry the load from the slabs and also the direct loads including their self-weight and the dead load of the structure. They carry the loads in bending which causes the beam to go in tension and compression. Although there have been many alternative methods proposed in the recent times for the analysis of the structures but due to the fact that their commercial applicability is yet to be proved, they have not been used much. Hence, Finite Element Method has just made a blip on radar. Presently, with the advancement in science and technology, the use of Finite Element Analysis (FEA) has increased. The higher processing power of the software, much accurate results, less time consumption adds to its benefits over other methods of analysis. In the present era, to study the behavior of the beam to impact loading, ANSYS modelling of the beam have been extensively used.

## II. INTRODUCTION TO ANSYS:

The ANSYS software has been designed as the Finite Element Analysis program with numerous capabilities ranging from simple linear static analysis to complex non-linear transient dynamic analysis.

Basically the analysis carried out in ANSYS has three basic distinct steps:

- Building the model
- Applying loads and obtaining the results
- Reviewing the results

### *2.1. BUILDING THE MODEL:*

Building the model is the first and foremost step required in ANSYS to carry out the analysis. Specifying the job name and the ANSYS title and then defining the element types, real constants, material properties and element geometry defines the whole process of generating the model. Building the finite element model is the most time consuming than any other part of ANSYS.

### *2.2. DEFINING ELEMENT TYPES:*

With each element having unique number and a prefix, identifying the element category, the element library for analysis in ANSYS contains more than 100 different element types.

### *2.3. DEFINING ELEMENT REAL CONSTANTS:*

The properties that depend on the element type like cross sectional properties of the beam element are called element real constants. E.g. real constants for BEAM3, moment of Inertia (IZZ), height, initial strain (ISTRN). Different elements of same type may attribute different element properties.

### *2.4. DEFINING MATERIAL PROPERTIES:*

The material properties which are required by most of the element types may be categorized into the following types depending upon the application:

- Linear or non-linear
- Isotropic, orthotropic and anisotropic
- Constant temperature or temperature dependent.

A definite material reference number is assigned to each set of material properties with element type and real constant. Material property table defines the relation between the material reference number and material property set.

## III. MESHING AND ITS IMPORTANCE

As the name depicts, the principal objective of the Finite Element Analysis is to bring out the calculations at finite or limited number of points and the results obtained are thus interpolated for the entire domain (surface/volume). As far as continuous object/boundary is concerned, it has infinite degree of freedom and it becomes hectic or sometimes impossible to solve the problem in this format. Hence by discretization or meshing, degrees of freedom are reduced from infinite to finite.

The precision of any Finite Element Analysis model is unswervingly associated to the finite element mesh being used. The computer aided design model (CAD) is subdivided into smaller domains with the aid of finite element mesh. These smaller domains are called elements. The set of equations nearly representing the governing equations of concern through a set of polynomial functions designated over each element, are solved over the minor domain. Once the mesh is refined by making the elements smaller and smaller, the reckoned solution approaches the true solution.

### *3.1. MESH REFINEMENT TECHNIQUE:*

Different techniques are employed for mesh refinement which include:

*a) Reducing the Element size:*

It is the facile mesh refinement technique in which the element sizes are condensed all-around the domain of modelling. Because of its simplicity, this approach is quiet attractive but the major drawback lies in the fact that regions where a locally finer mesh is needed, there is no privileged mesh refinement.

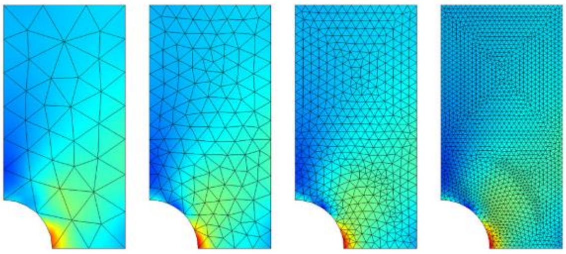


Fig.1: The stress in the plate with a hole solved with dissimilar element sizes

*b) Increasing the element order:*

By increasing the element order, similar mesh can be used but with dissimilar element orders and hence no re-meshing is needed. Re-meshing is extremely chronophagous for complex 3D geometries. The major drawback of this method is that computational requisites upsurge quicker than with other mesh refinement technique.

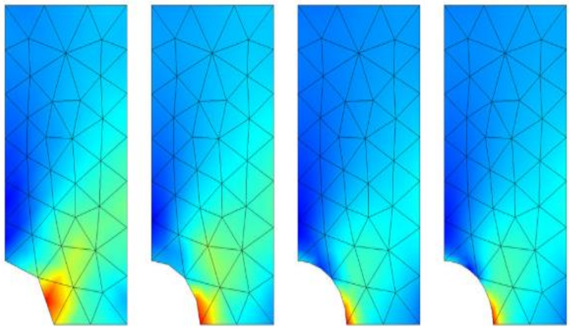


Fig.2: The identical finite element mesh but solved with different element orders

## IV. SIGNIFICANCE OF FINITE ELEMENT ANALYSIS:

The Finite Element Method is a numerical approach to find the contiguous solution of partial differential equations or it can also be defined as a portrayal of a body or a structure by a conglomeration of sub divisions called as finite elements. It was developed as a need of solving composite elasticity and structural analysis problems in Civil, Mechanical and Aerospace engineering.

In the structural simulation, FEM helps in producing stiffness and strength visualizations. The weight of the material and the structure cost is reduced with the help of FEM. The stress and strain distribution inside the body of the structure along with the comprehensive visualization is given by FEM. Many of the FE software are influential yet multifarious tool meant for specialized engineers with the training and education necessary to appropriately elucidate the results.

FEA is the computer model of the continuum that is stressed and scrutinized for particular results. A continuum has inestimable particles with unremitting variation of material properties. Hence it necessities to streamline to a finite size and is made up of aggregation of sub structures components and members. Discretization progression is essential to convert the complete structure to an accumulation of members for determining its reactions.

On the basis of speculations, the suitable constitutive model can be assembled. Aimed at the linear-elastic-static analysis of structure, the closing form of the equivalence will be made in the form of ***F=Kd,*** where F, k, d are nodal loads, global stiffness and nodal displacements respectively.

## V. RESULTS AND DISCUSSION:

A RCC beam is tested under impact loading using Explicit Dynamics platform by Finite Element Analysis (FEA) in ANSYS.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 8 Issue 07, July-2019

An impactor (solid punch) with high velocity approximately of approximately 15000mm/s and a mass of 12.519kg was allowed to strike the beam and the change in the parameters like the internal energy, kinetic energy, hourglass energy, contact energy, deformation of the beam [concrete as well as steel], equivalent stress, maximum principal stress and maximum principal elastic strain is analyzed.

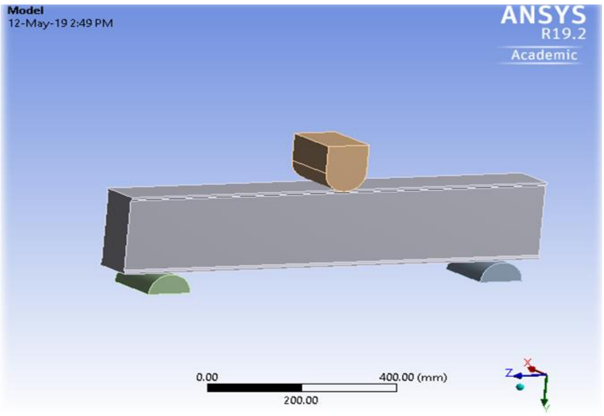



Fig.3: Model of the beam

### *5.1. GEOMETRY DESCRIPTION:*

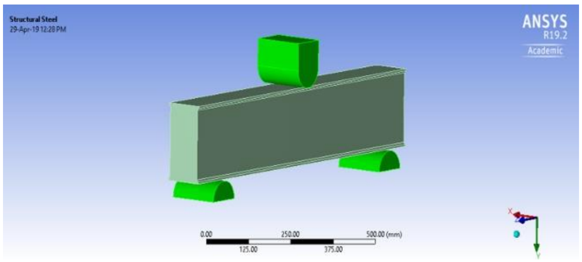



Fig. 4: Impactor and support

*Table 1: Geometrical properties of support and impactor*

| Object Name | *Solid - support 1* | *Solid - support 2* | *Solid - punch* |
|---|---|---|---|
| State | Meshed | | |
| **Graphics Properties** | | | |
| Visible | Yes | | |
| Transparency | 1 | | |
| **Definition** | | | |
| Suppressed | No | | |
| Stiffness Behavior | Flexible | | |
| Coordinate System | Default Coordinate System | | |
| Reference Temperature | By Environment | | |
| Reference Frame | Lagrangian | | |
| **Material** | | | |
| Assignment | Structural Steel | | |
| **Bounding Box** | | | |
| Length X | 150. mm | | |
| Length Y | 50. mm | | 119.01 mm |
| Length Z | 100. mm | | 98. mm |
| **Properties** | | | |
| Volume | 5.8905e+005 mm³ | | 1.5948e+006 mm³ |
| Mass | 4.624 kg | | 12.519 kg |
| Centroid X | -2.1188e-015 mm | -1.0594e-015 mm | 0. mm |
| Centroid Y | 128.83 mm | | -165.29 mm |
| Centroid Z | 100. mm | 900. mm | 499. mm |
| Moment of Inertia Ip1 | 3660.4 kg·mm² | | 22017 kg·mm² |
| Moment of Inertia Ip2 | 11487 kg·mm² | | 32536 kg·mm² |
| Moment of Inertia Ip3 | 9425.7 kg·mm² | | 36345 kg·mm² |
| **Statistics** | | | |
| Nodes | 396 | 352 | 924 |
| Elements | 260 | 220 | 700 |
| Mesh Metric | None | | |

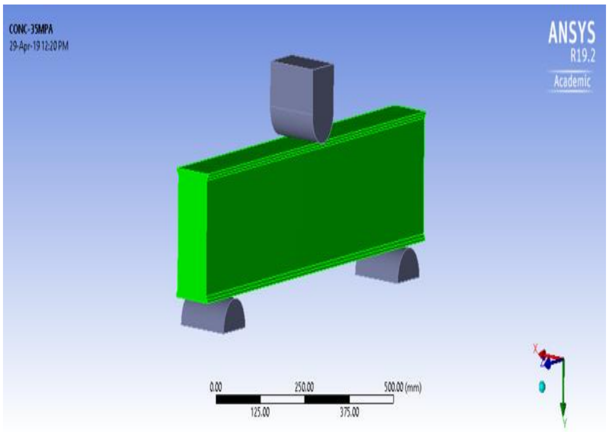



*Fig. 5: Concrete beam*

*Table 2: Geometrical properties of beam*

| Object Name | *Solid - concrete* |
|---|---|
| State | Meshed |
| **Graphics Properties** | |
| Visible | Yes |
| Transparency | 1 |
| **Definition** | |
| Suppressed | No |
| Stiffness Behavior | Flexible |
| Coordinate System | Default Coordinate System |
| Reference Temperature | By Environment |
| Reference Frame | Lagrangian |
| **Material** | |
| Assignment | CONC-35MPA |
| **Bounding Box** | |
| Length X | 110. mm |
| Length Y | 200. mm |
| Length Z | 1000. mm |
| **Properties** | |
| Volume | 2.015e+007 mm³ |
| Mass | 46.627 kg |
| Centroid X | 7.7803e-016 mm |
| Centroid Y | 1.0476e-015 mm |
| Centroid Z | 500. mm |
| Moment of Inertia Ip1 | 4.0431e+006 kg·mm² |
| Moment of Inertia Ip2 | 3.9251e+006 kg·mm² |
| Moment of Inertia Ip3 | 1.9699e+005 kg·mm² |
| **Statistics** | |
| Nodes | 9724 |
| Elements | 7772 |
| Mesh Metric | None |

### 5.2. MESHING DETAILS

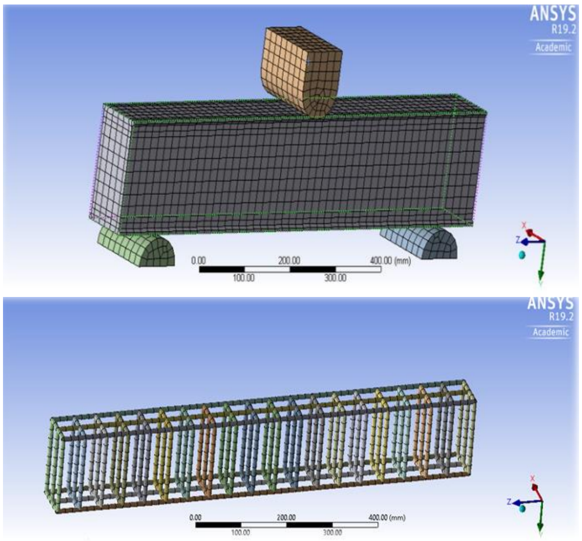



*Fig. 7: Meshing of reinforcement*

*Table 3: Meshing details*

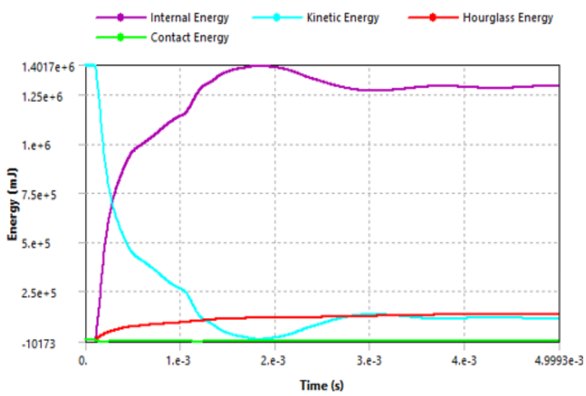
| | |
|---|---|
| Object Name | *Mesh* |
| State | Solved |
| **Display** | |
| Display Style | Use Geometry Setting |
| **Defaults** | |
| Physics Preference | Explicit |
| Element Order | Linear |
| Element Size | 15.0 mm |
| **Sizing** | |
| Use Adaptive Sizing | Yes |
| Resolution | Default (4) |
| Mesh Defeaturing | Yes |
| Defeature Size | Default |
| Transition | Slow |
| Span Angle Center | Coarse |
| Initial Size Seed | Assembly |
| Bounding Box Diagonal | 1076.8 mm |
| Average Surface Area | 30045 mm² |
| Minimum Edge Length | 5.0 mm |
| **Quality** | |
| Check Mesh Quality | Yes, Errors |
| Target Quality | Default (0.050000) |
| Smoothing | High |
| Mesh Metric | None |
| **Inflation** | |
| Use Automatic Inflation | None |
| Inflation Option | Smooth Transition |
| Transition Ratio | 0.272 |
| Maximum Layers | 5 |
| Growth Rate | 1.2 |
| Inflation Algorithm | Pre |
| View Advanced Options | No |
| **Advanced** | |
| Number of CPUs for Parallel Part Meshing | 4 |
| Straight Sided Elements | |
| Number of Retries | Default (4) |
| Rigid Body Behavior | Full Mesh |
| Triangle Surface Mesher | Program Controlled |
| Topology Checking | Yes |
| Pinch Tolerance | Please Define |
| Generate Pinch on Refresh | No |
| **Statistics** | |
| Nodes | 13974 |
| Elements | 10308 |

### 5.3. ENERGY SUMMARY

When the impactor is in motion, it possess very high **kinetic energy**. This kinetic energy possessed by the impactor is due to the very high velocity of the impactor which is approximately about 15000mm/s. The kinetic energy of the impactor is shown in the graph which is initially the maximum of all the energies as shown.

Before the collision takes place, **Internal energy** of the beam is very low as no load acts on it due to which no stresses or strains are developed in it. When the impactor strikes the beam with very high velocity, the kinetic energy of the impactor (which is very high) gets shifted to the beam and results in the sudden rise in the internal energy of the beam due to the generation of stresses and strains in the beam.

The duration of the time interval in which the whole process (i.e. Transfer of kinetic energy into the internal energy of the beam) takes place is very small i.e.; 1×**$10^{-3}$secs.**




*Graph 1: Energy Vs Time (Energy Summary)*

### 5.4. HOURGLASS ENERGY

- It is the work done by the forces to counterattack the **hourglass modes.**
- Hourglass modes are the non-physical, zero energy modes of distortion that yield zero strain and no stress. Hourglass modes ensue only in under-integrated [Single Integration Point], solid shell and thick shell elements [with single in-plane integration].
- These can affect the solution precision by prying with the structures true response and hence leads to erroneous stress, strain, and deflection and contact results.
- A high hourglass energy comparative to systems internal energy is good indicator that hourglassing is substantial and needs further quashing.

There are two ways to reduce or eliminate the hourglass energy i.e;

1. Fully integrated elements [but it has its own shortcomings]

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 8 Issue 07, July-2019

2. Refine the mesh in the regions that display hourglassing but it will lead to upsurge in the runtime of the software.
3. Point or edge loads or edge contact can stimulate hourglass modes. Dissemination the load over more elements is another way to reduce hourglass.

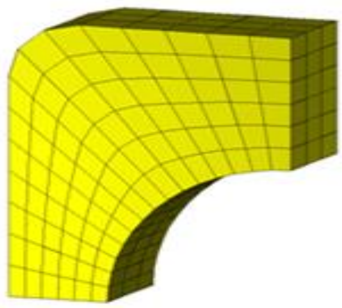


*Fig. 8: Mesh showing no hourglassing*

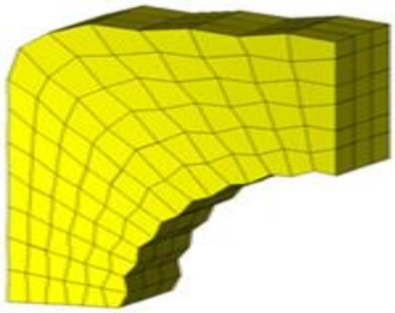


*Fig. 9: Mesh showing visible hourglassing*

5.5. TOTAL DEFORMATION IN CONCRETE:

While carrying out the explicit dynamic analysis of the beam and calculating the total deformation of concrete, it is found that a maximum deformation of **9.3605mm** occurs at a time step of **1.334e-003s.**

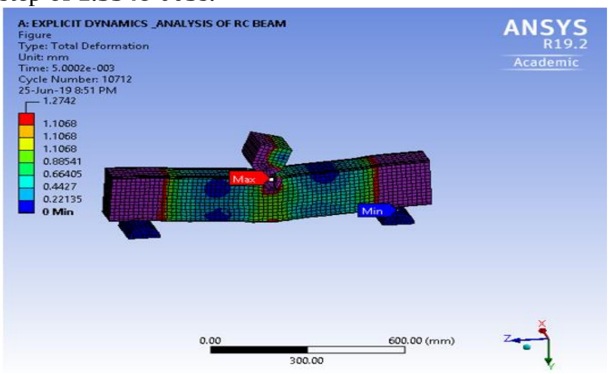
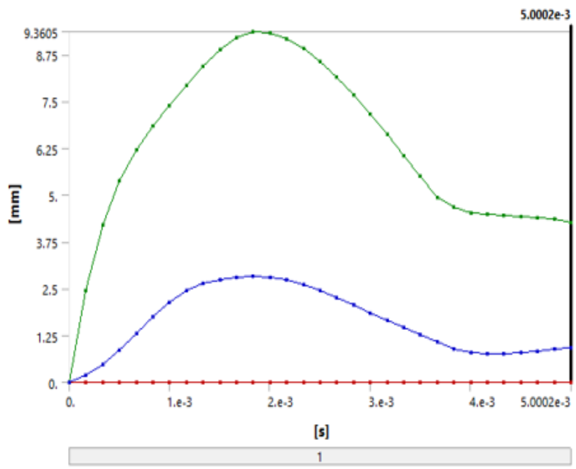



*Fig. 9: Concrete Deformation*

*Table 4: Concrete deformation.*

| Time [s] | Minimum [mm] | Maximum [mm] | Average [mm] |
|---|---|---|---|
| 1.1755e-038 | | | |
| 1.6689e-004 | | 2.4475 | 0.17628 |
| 3.3363e-004 | | 4.2016 | 0.46396 |
| 5.0037e-004 | | 5.3891 | 0.86342 |
| 6.671e-004 | | 6.2196 | 1.3101 |
| 8.3335e-004 | | 6.8332 | 1.7524 |
| 1.0001e-003 | | 7.3843 | 2.1308 |
| 1.1668e-003 | | 7.9192 | 2.4545 |
| 1.3335e-003 | | 8.4371 | 2.63 |
| 1.5001e-003 | | 8.8911 | 2.7312 |
| 1.6668e-003 | | 9.2101 | 2.8072 |
| **1.8334e-003** | | **9.3605** | 2.8369 |
| 2.0001e-003 | | 9.3435 | 2.8055 |
| 2.1668e-003 | | 9.1829 | 2.7242 |
| 2.3334e-003 | | 8.9166 | 2.6101 |
| 2.5001e-003 | 0. | 8.5709 | 2.4597 |
| 2.6667e-003 | | 8.147 | 2.2661 |
| 2.8334e-003 | | 7.665 | 2.0601 |
| 3.e-003 | | 7.1506 | 1.8595 |
| 3.1671e-003 | | 6.6089 | 1.6623 |
| 3.3338e-003 | | 6.0524 | 1.466 |
| 3.5004e-003 | | 5.4943 | 1.2705 |
| 3.667e-003 | | 4.9416 | 1.0797 |
| 3.8337e-003 | | 4.6767 | 0.90345 |
| 4.0003e-003 | | 4.5364 | 0.79275 |
| 4.167e-003 | | 4.476 | 0.75617 |
| 4.3336e-003 | | 4.4471 | 0.76028 |
| 4.5002e-003 | | 4.4281 | 0.78756 |
| 4.6669e-003 | | 4.4057 | 0.82887 |
| 4.8335e-003 | | 4.3611 | 0.87757 |
| 5.0002e-003 | | 4.2672 | 0.93091 |



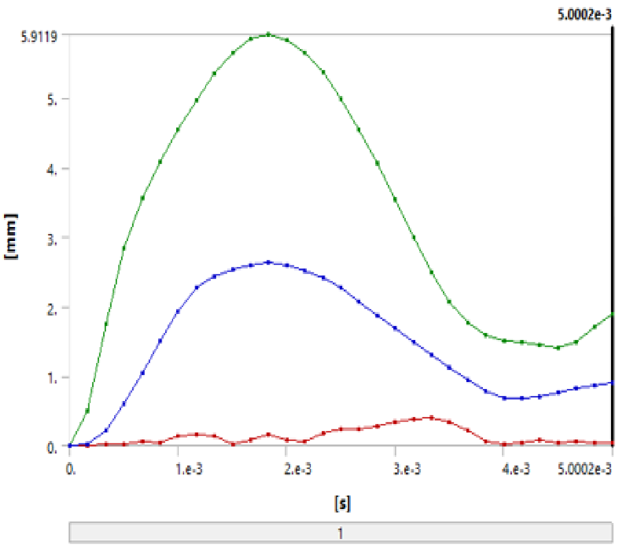

*Graph 2: Deformation vs Time in Concrete*

5.6. TOTAL DEFORMATION IN STEEL

The maximum deformation in the reinforcement was found to be **5.9119mm** which occurs at a time step of **2.00e-003s**.

*Table 5: Deformation in reinforcement*

| Time [s] | Minimum [mm] | Maximum [mm] | Average [mm] |
|---|---|---|---|
| 1.1755e-038 | | | |
| 1.6689e-004 | 0. | 0.49377 | 1.5839e-002 |
| 3.3363e-004 | 1.3019e-002 | 1.7566 | 0.21738 |
| 5.0037e-004 | 2.5898e-002 | 2.8393 | 0.59465 |
| 6.671e-004 | 6.6721e-002 | 3.5642 | 1.0504 |
| 8.3335e-004 | 3.4154e-002 | 4.0835 | 1.5179 |
| 1.0001e-003 | 0.1403 | 4.543 | 1.9236 |
| 1.1668e-003 | 0.16875 | 4.9742 | 2.2701 |
| 1.3335e-003 | 0.13827 | 5.3514 | 2.4427 |
| 1.5001e-003 | 2.2907e-002 | 5.6502 | 2.5308 |
| 1.6668e-003 | 7.8122e-002 | 5.8466 | 2.6022 |
| 1.8334e-003 | 0.15372 | 5.9119 | 2.6308 |
| 2.0001e-003 | 8.7252e-002 | 5.8409 | 2.5976 |
| 2.1668e-003 | 6.9775e-002 | 5.6529 | 2.5181 |
| 2.3334e-003 | 0.17308 | 5.3649 | 2.411 |
| 2.5001e-003 | 0.25048 | 4.9897 | 2.2658 |
| 2.6667e-003 | 0.24713 | 4.5496 | 2.075 |
| 2.8334e-003 | 0.27344 | 4.0662 | 1.8735 |
| 3.e-003 | 0.34674 | 3.5388 | 1.6793 |
| 3.1671e-003 | 0.38772 | 2.9939 | 1.4909 |
| 3.3338e-003 | 0.39679 | 2.4945 | 1.3053 |
| 3.5004e-003 | 0.34209 | 2.0812 | 1.1217 |
| 3.667e-003 | 0.22131 | 1.7755 | 0.94463 |
| 3.8337e-003 | 5.035e-002 | 1.597 | 0.78348 |
| 4.0003e-003 | 2.9006e-002 | 1.517 | 0.6917 |
| 4.167e-003 | 3.9857e-002 | 1.4785 | 0.68015 |
| 4.3336e-003 | 8.1032e-002 | 1.4483 | 0.71352 |
| 4.5002e-003 | 4.2193e-002 | 1.4081 | 0.76509 |
| 4.6669e-003 | 5.5311e-002 | 1.4906 | 0.82113 |
| 4.8335e-003 | 4.4865e-002 | 1.7169 | 0.87133 |
| 5.0002e-003 | 4.4356e-002 | 1.8818 | 0.91209 |




*Graph 3: Deformation Vs Time in Steel*

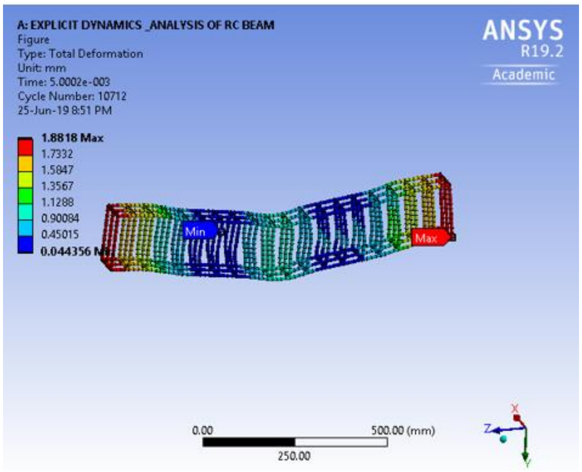



*Fig. 10: Reinforcement deformation*

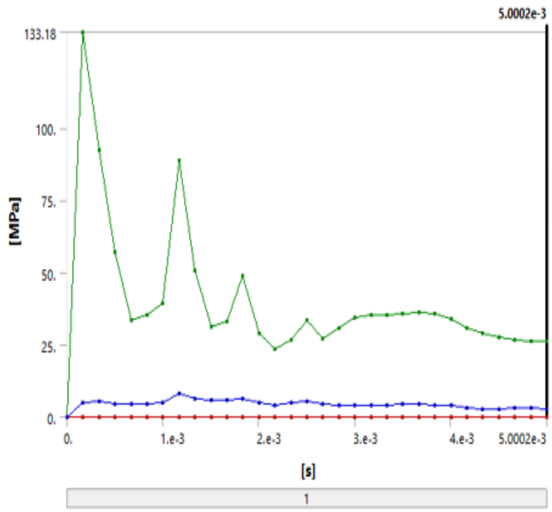
5.7. EQUIVALENT STRESS IN THE BEAM

Equivalent stress, also known as von misses stress, characterizes any random three dimensional stress state as a single positive stress value. Equivalent stress is a subset of maximum equivalent stress failure theory which is used to envisage yielding of ductile material.

The relation between the equivalent stress and principal stress is given as:

$$\sigma_e = \left[ \frac{(\sigma_1-\sigma_2)^2+(\sigma_2-\sigma_3)^2+(\sigma_3-\sigma_1)^2}{2} \right]^{1/2}$$

From the analysis carried out on the beam using explicit dynamics approach, it was found that maximum equivalent stress of **133.18 MPa** occurs in the beam at a time step of **1.6689e-004s**




*Graph 4: Equivalent Stress Vs Time*

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 8 Issue 07, July-2019

*Table 6: Equivalent Stress in Beam*

| Time [s] | Minimum [MPa] | Maximum [MPa] | Average [MPa] |
|---|---|---|---|
| 1.1755e-038 | | | |
| **1.6689e-004** | | **133.18** | 4.9451 |
| 3.3363e-004 | | 92.527 | 5.3973 |
| 5.0037e-004 | | 57.302 | 4.4208 |
| 6.671e-004 | | 33.415 | 4.4337 |
| 8.3335e-004 | | 35.445 | 4.5824 |
| 1.0001e-003 | | 39.385 | 4.7856 |
| 1.1668e-003 | | 88.959 | 7.9394 |
| 1.3335e-003 | | 50.642 | 6.568 |
| 1.5001e-003 | | 31.382 | 5.7271 |
| 1.6668e-003 | | 33.002 | 6.0831 |
| 1.8334e-003 | | 48.707 | 6.3707 |
| 2.0001e-003 | | 29.128 | 4.7957 |
| 2.1668e-003 | | 23.682 | 4.1424 |
| 2.3334e-003 | | 26.558 | 4.8214 |
| 2.5001e-003 | 0. | 33.536 | 5.2755 |
| 2.6667e-003 | | 27.088 | 4.4486 |
| 2.8334e-003 | | 30.722 | 4.043 |
| 3.e-003 | | 34.394 | 3.92 |
| 3.1671e-003 | | 35.147 | 4.035 |
| 3.3338e-003 | | 35.199 | 4.2242 |
| 3.5004e-003 | | 35.89 | 4.3823 |
| 3.667e-003 | | 36.42 | 4.3043 |
| 3.8337e-003 | | 35.709 | 4.0839 |
| 4.0003e-003 | | 33.75 | 3.8556 |
| 4.167e-003 | | 30.9 | 3.361 |
| 4.3336e-003 | | 29.045 | 2.9148 |
| 4.5002e-003 | | 27.61 | 2.8983 |
| 4.6669e-003 | | 26.502 | 3.1099 |
| 4.8335e-003 | | 26.337 | 3.1394 |
| 5.0002e-003 | | 26.251 | 2.9189 |

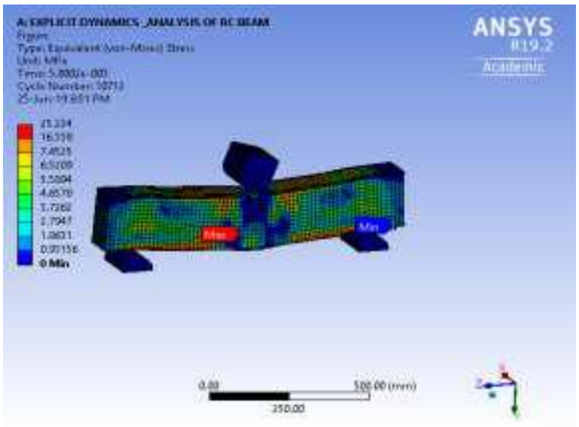



*Fig. 11: Equivalent (von-misses stress) stress*

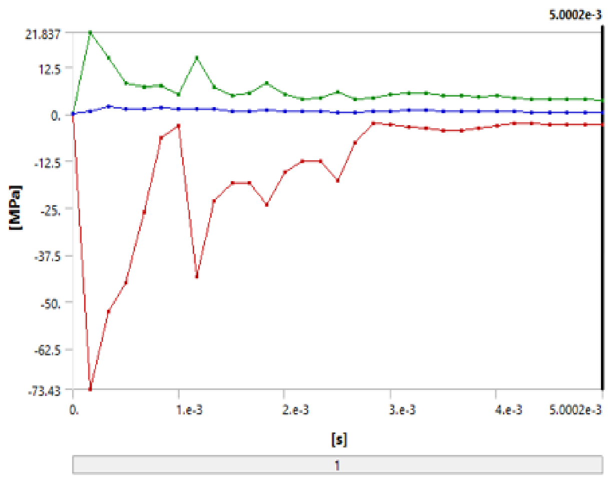
## 5.8. MAXIMUM PRINCIPAL STRESS

When the shear stress approaches zero at an angle ϴ, the normal stresses calculated at this angle are known as principal stress. The normal stresses calculated at the maximum values are known as major principal stresses and those calculated from minimum values are known as minor principal stresses.

From the analysis results, the maximum principal stress of **21.837 MPa** was found to occur in beam at a time step of **1.6689e-004s** and a minimum principal stress of **-73.43 MPa** was found at a time step of **1.6689e-004s.**

*Table 7: Maximum Principal Stress*

| Time [s] | Minimum [MPa] | Maximum [MPa] | Average [MPa] |
|---|---|---|---|
| 1.1755e-038 | | | |
| 1.6689e-004 | **-73.43** | **21.837** | 0.79523 |
| 3.3363e-004 | -52.712 | 14.981 | 1.9348 |
| 5.0037e-004 | -44.883 | 8.1526 | 1.4363 |
| 6.671e-004 | -26.141 | 7.1107 | 1.3134 |
| 8.3335e-004 | -6.2984 | 7.5886 | 1.6624 |
| 1.0001e-003 | -3.1053 | 5.2296 | 1.4059 |
| 1.1668e-003 | -43.165 | 14.994 | 1.4693 |
| 1.3335e-003 | -23.239 | 7.0972 | 1.4416 |
| 1.5001e-003 | -18.276 | 5.0279 | 0.86425 |
| 1.6668e-003 | -18.356 | 5.7908 | 0.68583 |
| 1.8334e-003 | -24.166 | 8.2125 | 1.0414 |
| 2.0001e-003 | -15.459 | 5.225 | 0.83428 |
| 2.1668e-003 | -12.363 | 3.9835 | 0.68624 |
| 2.3334e-003 | -12.63 | 4.3926 | 0.66873 |
| 2.5001e-003 | -17.817 | 5.8826 | 0.55272 |
| 2.6667e-003 | -7.7185 | 3.9377 | 0.53516 |
| 2.8334e-003 | -2.3099 | 4.4946 | 0.66521 |
| 3.e-003 | -2.8621 | 5.1996 | 0.74577 |
| 3.1671e-003 | -3.324 | 5.6002 | 0.9511 |
| 3.3338e-003 | -3.9095 | 5.638 | 1.0079 |
| 3.5004e-003 | -4.4688 | 5.0875 | 0.90841 |
| 3.667e-003 | -4.5199 | 5.041 | 0.87913 |
| 3.8337e-003 | -3.8166 | 4.594 | 0.74366 |
| 4.0003e-003 | -3.2405 | 5.0997 | 0.74051 |
| 4.167e-003 | -2.4684 | 4.3835 | 0.69689 |
| 4.3336e-003 | -2.5571 | 4.1575 | 0.46786 |
| 4.5002e-003 | -2.6751 | 3.9729 | 0.3929 |
| 4.6669e-003 | -2.7681 | 4.1434 | 0.45384 |
| 4.8335e-003 | -2.8138 | 3.8798 | 0.37994 |
| 5.0002e-003 | -2.7869 | 3.8231 | 0.36514 |




*Graph 5: Maximum principal Stress Vs Time.*

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 8 Issue 07, July-2019

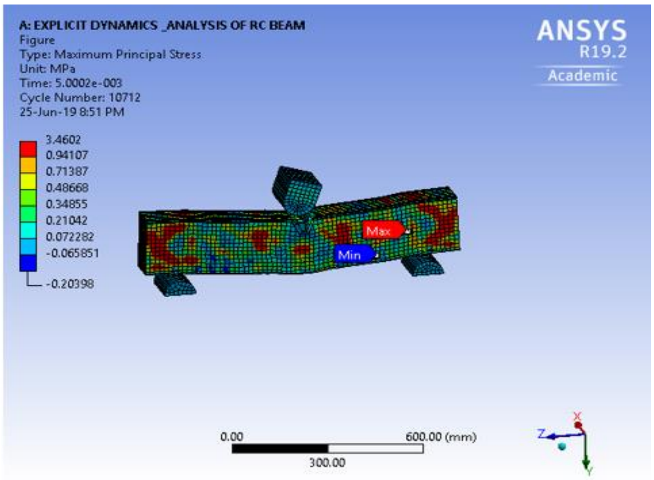


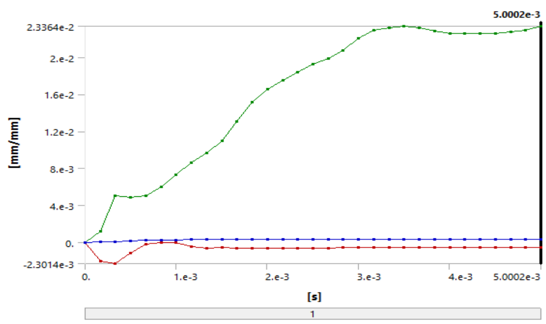

*Fig. 12: maximum principal stress*

5.9. MAXIMUM PRINCIPAL ELASTIC STRAIN:

From the analysis of the beam it is clear that maximum principal elastic strain of **2.3364e-002 mm/mm** occurs at a time step of **5.0004e003s**.

*Table 8: Maximum principal elastic strain*

| Time [s] | Minimum [mm/mm] | Maximum [mm/mm] | Average [mm/mm] |
|---|---|---|---|
| 1.1755e-038 | | | |
| 1.6689e-004 | -2.0317e-003 | 1.2263e-003 | 3.0433e-005 |
| 3.3363e-004 | -2.3014e-003 | 5.0084e-003 | 9.847e-005 |
| 5.0037e-004 | -1.1509e-003 | 4.8279e-003 | 1.6888e-004 |
| 6.671e-004 | -1.7934e-004 | 5.0368e-003 | 2.0876e-004 |
| 8.3335e-004 | -2.7346e-005 | 6.024e-003 | 2.4764e-004 |
| 1.0001e-003 | -2.0544e-005 | 7.2617e-003 | 2.7205e-004 |
| 1.1668e-003 | -4.3478e-004 | 8.5677e-003 | 2.9721e-004 |
| 1.3335e-003 | -6.4946e-004 | 9.6322e-003 | 2.8703e-004 |
| 1.5001e-003 | -5.9506e-004 | 1.1003e-002 | 2.7559e-004 |
| 1.6668e-003 | -6.1877e-004 | 1.3103e-002 | 2.8556e-004 |
| 1.8334e-003 | -6.0581e-004 | 1.5167e-002 | 3.1005e-004 |
| 2.0001e-003 | -6.4597e-004 | 1.655e-002 | 3.1558e-004 |
| 2.1668e-003 | -6.083e-004 | 1.7489e-002 | 3.1526e-004 |
| 2.3334e-003 | -6.1832e-004 | 1.8352e-002 | 3.0605e-004 |
| 2.5001e-003 | -6.2722e-004 | 1.9245e-002 | 2.9496e-004 |
| 2.6667e-003 | -6.2476e-004 | 1.9906e-002 | 2.896e-004 |
| 2.8334e-003 | -5.3024e-004 | 2.0763e-002 | 2.9582e-004 |
| 3.e-003 | -5.3763e-004 | 2.2042e-002 | 3.0951e-004 |
| 3.1671e-003 | -5.4431e-004 | 2.2915e-002 | 3.2314e-004 |
| 3.3338e-003 | -5.4394e-004 | 2.3227e-002 | 3.2838e-004 |
| **3.5004e-003** | -5.4491e-004 | **2.3364e-002** | 3.2606e-004 |
| 3.667e-003 | -5.4671e-004 | 2.3158e-002 | 3.203e-004 |
| 3.8337e-003 | -5.4896e-004 | 2.2843e-002 | 3.1466e-004 |
| 4.0003e-003 | -5.497e-004 | 2.2614e-002 | 3.1021e-004 |
| 4.167e-003 | -5.5266e-004 | 2.2552e-002 | 3.0515e-004 |
| 4.3336e-003 | -5.5408e-004 | 2.2544e-002 | 2.9715e-004 |
| 4.5002e-003 | -5.5419e-004 | 2.2599e-002 | 2.9064e-004 |
| 4.6669e-003 | -5.5335e-004 | 2.2715e-002 | 2.8769e-004 |
| 4.8335e-003 | -5.5168e-004 | 2.2965e-002 | 2.87e-004 |
| 5.0002e-003 | -5.5204e-004 | 2.3351e-002 | 2.9043e-004 |




*Graph 6: Maximum elastic strain vs time*

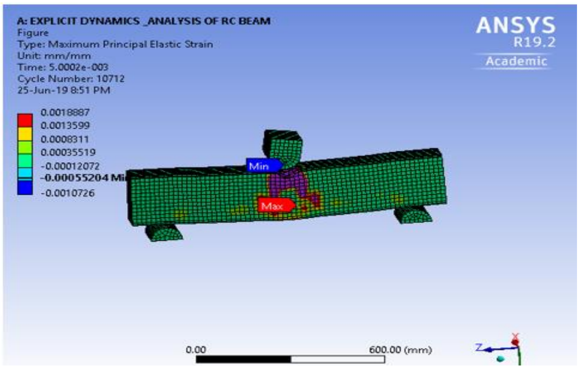



*Fig. 13: Maximum Principal Elastic Strain*

## VI. CONCLUSION

Based on the ANSYS modelling and the analysis carried on the beam, the following conclusions were drawn;

- The behavior of reinforced concrete beam is scrutinized using finite element method. The parameters used in this study are impact loading (Explicit Dynamics) and variation of energy.
- Reinforced concrete beam can be modelled and analyzed using ANSYS of version 19.2 software and accurate results can be obtained.
- From the analysis of the beam under various impact loads it was found that greater the kinetic energy of the impactor, greater will be the internal energy in the beam after the impactor gets strike and hence greater will be the stresses and strains developed in the beam.
- Hourglass energy can affect the solution correctness by snooping with the structures true response and hence leads to erroneous stress, strain, and deflection and contact results and hence should be kept as minimum as possible.