Chapter 5

SIMULATIONS OF EXPERIMENTAL WORK PERFORMED ON 3D C-SiC COMPOSITE SPECIMENS BY USING ANSYS

5.1 Experimental configurations:

5.1.1 Impact test:

The configuration used in experiments is shown in Fig: 5.1 where the striker of nose radius $r$, traveling at a velocity $v$, impacts the specimen from a height $h$, at a distance of 22 mm from V-notch along the length of the specimen. A V-notch of root radius 0.25 mm with included angle of $45^\circ$ having a depth of 2.54 mm is provided following the ASTM D 256 standards for the Izod impact test. The configuration is described conveniently in terms of rectangular coordinates.

Fig: 5.1 Impact test specimen with loading and boundary conditions
The geometric model is created using ANSYS LS-DYNA software. A 3D structural solid 164 element is used to mesh the geometry of the striker as well as the specimen. Solid 164 element is having 8 nodes with the following degrees of freedom at each node: translations, velocities, and accelerations in the nodal x, y and z directions.

3D solid modeling is used to generate the specimen which then is meshed with solid 164 element shown in Fig: 5.2. A refined mesh is obtained after convergence check with a total number of 880 elements having 1260 nodes as shown in Fig: 5.3.

Fig: 5.2 solid 164 element
The impact process is simulated on the computer by following a transient analysis. The objective of this work is to demonstrate the explicit dynamics using ANSYS LS-DYNA for complex contact dynamics situation which simulates Izod impact test of experimentation.

During the free fall stage, the striker is simply accelerating due to gravity. The analysis is started when the striker is 0.5 m above the specimen in order to save CPU time. The initial velocity of 3 ms\(^{-1}\) is applied to simulate the process. This velocity is an approximation obtained by using \( V = (2gh)^{1/2} \) where g is acceleration due to gravity and h is displacement. Air friction is assumed to be neglected.
The striker is constrained to have translations in y – direction (1dof). It is restrained to have translations in x, z directions and rotations about x, y, z directions. The deformation obtained from ANSYS is 3.92 mm as shown in Fig: 5.4 where as the actual experimental value for the specimen is 3.95 mm. The percentage variation is 0.75.

The stress distribution is shown in Fig: 5.5. The maximum stress obtained in y-direction is 13.95 MPa. The experimental value is 14.2 MPa. It is observed that there is a variation of 1.8 %. The impact strength obtained for the sample specimen # 4 by FEA analysis is 27.36 kJ/m² which is 2 % more than the experimental value of 26.82 kJ/m². Comparison of experimental and FEA results is shown in Fig: 5.6.

Fig: 5.4 Deformation of the specimen in mm
Fig: 5.5 Stress distribution in MPa.

Fig: 5.6 Comparison of experimental and FEA results of Impact Strength
5.1.2 **Flexural test:**

The specimen geometry for 3 point bend test as per ASTM C 1341 standards is shown in Fig: 5.7. The specimen is held in the testing machine as a simply supported beam and load is gradually applied at the centre. When the applied load reaches ultimate value the specimen breaks and subsequently the load falls to zero.

![Flexural test specimen with loading and boundary conditions](image)

The finite element model is generated using ANSYS 11.0 software. A 3D solid 45 brick element with 8 nodes is used to mesh the geometry of the specimen. Solid 45 element has 3 dofs. A refined mesh is obtained with 1560 elements and 2376 nodes which is shown in Fig: 5.8.
The computer simulations of flexural test are performed by choosing the ultimate loads recorded in the test. The deformation obtained by computer simulation of flexural test is 2.58 mm as shown in Fig: 5.9. It appears to be very close to the experimental value of 2.5 mm. The variation is 3 %.

The flexural stress distribution is shown in Fig: 5.10. The maximum flexural strength obtained by computer analysis for specimen # 5 is 240.17 MPa where as its experimental value is 230.3 MPa. The variation is 4.3 % which is within acceptable range.

Fig: 5. 8 Finite Element model of the test specimen
Fig: 5.9 Deformation of the flexural specimen in mm

Fig: 5.10 Flexural stress distribution in MPa
The shear stress distribution is shown in Fig: 5.11. The maximum shear stress obtained from this analysis is observed to be 31.73 MPa and the corresponding experimental value is 30.5 MPa. The percentage variation is 4.0 which is within the acceptable range. Comparison of experimental and FEA results of Flexural Strength is shown in Fig: 5.12.

Fig: 5.11 Shear stress distribution in MPa
5.1.3 Tensile test:

The geometrical data input to the computer is taken from the tensile test configuration according to ASTM C 1275 standards. The loading and boundary conditions are shown in Fig: 5.13. The specimen is fixed in the testing machine and the movable jaw is adjusted for the gauge length of 25 mm. The tensile load is gradually applied till the specimen is broken at the average max. values of 1.26-1.62 kN. The load then falls to zero.
The Finite Element model is created by using ANSYS 11.0 software. The specimen is meshed with the 3D solid 45 brick element which has 3 dofs. Fig: 5.14 shows the finite element model of the tensile test specimen obtained with 9000 elements and 11466 nodes.
The test process is simulated on the computer by running ANSYS program. One end of the specimen is fully restrained and the other end is constrained to have translations along the principal material direction. The computer simulations are performed by applying a gradual load. The analysis is completed when the load reaches a maximum value of 1.26-1.62 kN for different specimens. The deformation obtained in ANSYS is 0.23 mm as shown in Fig: 5.15. This is very near to the experimental value of 0.25 mm.
The tensile strength distribution for the specimen is shown in Fig: 5.16. The tensile strength obtained for specimen #1 from this analysis is 71.735 MPa. Its experimental value is observed to be 70.2 MPa. The variation in the two values is noticed to be 2 %. Comparison of experimental and FEA results of Tensile Strength is shown in Fig: 5.17.
Fig: 5.16 Tensile strength distribution in MPa
Fig: 5.17 Comparison of experimental and FEA results of Tensile Strength