CHAPTER 4

DEVELOPMENT OF FINITE ELEMENT MODEL

4.1 GENERAL

The difficulties in modelling the bi-material interface in brittle-matrix-composites (BMCs) have been discussed earlier. Finite element studies of BMCs are limited. Laird II and Kennedy [1991] used discrete spring slider elements (ANSYS) to model the interfacial connectivity between the matrix and whisker in the finite element analysis of whisker reinforced ceramic-matrix-composites (CMCs). This approach models the connectivity between the whisker and matrix at discrete nodal points only. As a result, the chances of overlapping of the nodes along the interface is not eliminated. Further, this type of idealisation of interface considers only the shear failure of the interface and cannot consider the tensile (normal) failure of the interface which is also possible in BMCs. Therefore with this approach, modelling the mechanics at the interface realistically is difficult. Rao&Mukherjee (1995) presented the analysis of CMCs using plane strain interface elements. However, the plane strain analysis can not model the cylindrical fibre/matrix interface that exists in BMCs realistically.

In the present investigation, a micromechanical finite element model is developed to analyse the BMCs, taking the bi-material interface into effect. The matrix, and the fibre are modelled by eight node isoparametric axi-symmetric quadrilateral elements. The fibre/matrix interface is modelled by six noded isoparametric axi-symmetric interface elements formulated in the previous chapter. As elements of compatible shape functions are used to model all the components, i.e., the fibre, matrix and their interface, the interaction between all the constituents is modelled appropriately. Also, the bi-material interface is
modelled all along the interface length and the model is capable of considering both shear and normal failures of the interface. The possibility of overlapping is also effectively eliminated. The developed finite element model is then used for deriving the stress-strain response of SiC (matrix)/SiC (fibre) brittle matrix composite. This stress-strain response has been compared with the experimental results of Lamon et al., (1993). The details of the model development and the validation have been presented in the following sections.

4.2 THE AXI-SYMMETRIC FINITE ELEMENT MODEL

Developing an adequate finite element model (FE model) for the analysis of brittle-matrix-composites involves modelling of:

- the matrix
- fibre and
- the interface between the two materials (see Fig. 4.1).

![Diagram of Bi-material Interfaces](image)

**Fig. 4.1:** Longitudinal section of a typical BMC showing the fibre matrix and their interfaces.

Modelling the mechanics of the bi-material interface in BMCs is rather complex as mentioned earlier. The complexity is due to the possible relative displacements at the bi-
material interface. Further, if the interface is in compression, a frictional force between the two material exists. This frictional grip between the matrix and the fibre resists the sliding of the fibre against a cracked matrix. If this frictional force is directly considered in the analysis, the system becomes non-conservative and the global stiffness matrix in the finite element analysis becomes unsymmetric. Such unsymmetric matrices pose difficulties in the solution routines, resulting in numerical instability. On the other hand, if the interface is in tension, the contact between the matrix and the fibre ceases to exist and hence there will not be any frictional force. Hence it has been rather difficult to model the bi-material interface in the finite element analysis (FEA) of brittle-matrix-composites. In the present finite element model, the matrix and the fibre are modelled by standard eight node isoparametric axi-symmetric quadrilateral elements (see Fig. 4.2).

However, to model the fibre/matrix interface, the mechanics of behaviour at the interface are treated to be similar to those of rock joints, where relative displacements occur across a thin discontinuity. With this treatment, it is proposed to model the bi-material interface using isoparametric axi-symmetric interface element formulated in chapter 3.
Thus, in the present axi-symmetric finite element model of the BMCs, the matrix and fibre are modelled by standard eight node isoparametric axi-symmetric quadrilateral elements and the fibre/matrix interface is modelled by the six noded isoparametric axi-symmetric interface element. These isoparametric interface elements model the connectivity between the matrix and fibre allowing slip when the interface shear stress exceeds the interface shear strength ($\tau_s$) of the composite. At each load increment, iterative finite element analysis (FEA) has to be carried out to account for shear failure or separation at the interface. Initially, the value of shear stiffness ($K_s$) is set to a very large value to avoid any non-slip displacement. When the interface shear stress ($\sigma_s$) at a point exceeds the interface strength, the shear stiffness ($K_s$) of that point is set to zero, thus allowing slip. If the normal stress ($\sigma_n$) at the interface is compressive there is a possibility of overlapping of the nodes of the adjacent elements. To avoid this, the value of normal stiffness ($K_n$) of the interface elements is set to be about thousand times the maximum value of Young's modulus of the bordering elements, in compression. When the normal stress ($\sigma_n$) of a point exceeds the interface normal strength, the normal stiffness ($K_n$) and the shear stiffness ($K_s$) of that point is set to zero to allow debonding at the interface. Thus the algorithm of the interface element takes the value of interface stiffness as:

- $K_s = 0$, if $\sigma_s >$ interface shear strength (slip condition).
- $K_n = 10^3 \times E_m$, if $\sigma_n < 0$ (overlapping condition).
- $K_s = 0$, and $K_n = 0$ if $\sigma_n >$ interface normal strength (debonding condition).

The iterative finite element analysis has to be carried out at each load increment and the stresses at the interface have to be monitored. To start with, the nature of the normal stress ($\sigma_n$) at the interface is not known. Hence the value of the normal stiffness ($K_n$) may be chosen assuming that the interface normal stress ($\sigma_n$) is tensile. A wrong assumption of the nature of the normal stress ($\sigma_n$) does not affect the final solution but
only increases the number of iterations. This has been thoroughly checked and confirmed by solving a problem with different starting values for the normal stiffness ($K_n$). Though this type of FE solution is computationally expensive, it is still a powerful tool for performing numerical experiments and sensitivity studies on BMCs in preference to the expensive and time consuming experimentation.

### 4.3 ADVANTAGES OF THE PRESENT FINITE ELEMENT MODEL

In the present finite element model, the bi-material interface is modelled by isoparametric axi-symmetric interface elements as described above. This isoparametric axi-symmetric interface elements with quadratic variation of both geometry and slip use the relative displacements between the adjacent matrix and fibre elements as its degrees of freedom. Therefore, the use of interface element for modelling the interface does not introduce any additional degrees of freedom in the problem. Also, the ill-conditioning problems that are commonly associated with interface elements are minimised with this formulation. Further, the present finite element model of BMCs is superior to other discrete spring models [Laird II and Kennedy, 1991] because in the displacement based finite element formulation of isoparametric interface element, the compatibility conditions are satisfied along the entire length of the interface. Moreover, the formulation of isoparametric interface element is an integral part of the finite element model. Also, the chances of overlapping of adjacent nodes along the interface when it is in compression are totally avoided in this approach. The finite element model is capable of considering both shear and normal failures of the interface.

### 4.4 VALIDATION OF THE FE MODEL

The finite element axi-symmetric model formulated above has to be validated against known results. In the present investigation, the validation of the model is carried out by deriving the stress-strain response of SiC/SiC composite and comparing the same with the experimentally derived response. Experiments for deriving the tensile stress-strain responses of these class of composites are rather complicated and expensive.
Further, they involve preparation of test specimens of very small sizes [Lamon et al., 1993]. Due to this, the preparation of the test specimen takes quite long time, some times it may take more than a month for preparing one test specimen. In addition to this, the reproducibility of the test specimens is generally very difficult to achieve. This creates sampling problems during the analysis of the experimental results. Figure 4.3 highlights these sampling difficulties, for the case of SiC (matrix)/SiC (fibre) composite.

![Stress-strain response graph](image)

**Fig. 4.3: Stress-strain responses obtained on seven different micro-composite tensile test specimens of SiC/SiC composite.**

The fig. 4.3 shows the tensile stress-strain responses of SiC (matrix)/SiC (fibre) obtained from the tensile test conducted on seven microcomposite specimen presented by Lamon et al., (1993). It can be seen from the figure that, the seven different specimen tested produced seven different responses. It can be observed that, no two curves are
matching. This highlights the difficulties in the reproducibility of the test specimens. Due to all these reasons, it will be very useful if the experimental responses can be simulated through numerical procedure. An attempt is made here to simulate the stress-strain response of SiC/SiC composite using the finite element model developed in the present work. This is achieved by constructing a finite element model of the microcomposite tensile test specimen recommended by Lamon et al., (1993). The details of the modelling and the results of the numerical simulations are presented in the following sections.

Material properties used in this study are given in Table 4.1 [Lamon et al., 1993]

<table>
<thead>
<tr>
<th>Property</th>
<th>Matrix (SiC)</th>
<th>Fibre (SiC)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Young's modulus (GPa)</td>
<td>180.0</td>
<td>210.0</td>
</tr>
<tr>
<td>Poisson's ratio</td>
<td>0.22</td>
<td>0.22</td>
</tr>
<tr>
<td>Tensile strength (MPa)</td>
<td>300</td>
<td>2.00 Gpa</td>
</tr>
<tr>
<td>Radius</td>
<td>-</td>
<td>4.2 μm</td>
</tr>
</tbody>
</table>

In axisymmetric formulation, the cylindrical fibre geometry is recognised. Due to this, the formulation should be able to model the geometry more realistically. For this purpose, the axi-symmetric finite element model developed will be used for simulating the tensile stress-strain test on SiC/SiC composite. An axi-symmetric finite element model of the microcomposite tensile test specimen of SiC/SiC has been constructed and depicted in Fig. 4.4.
Due to symmetry, the fibre must have the same axial ($z$) displacement at $z = 1680$ μm. This is achieved by giving the same equation number for the axial displacement of all the fibre nodes at $z = 1680$ μm. Also, the nodes along the axis of revolution have been locked in the radial direction, to impose the symmetry along the axis of revolution. The fibre nodes along the bottom axis ($z = 0$) have been locked. A finite element discretisation of the model has been carried out. This FE mesh consists of 160 matrix and 80 fibre eight node axi-symmetric quadrilateral elements. The fibre/matrix interface has been modelled by 80 six-node isoparametric axi-symmetric interface elements. Three interface elements are used at the bottom axis of symmetry ($z = 0$) as control elements. When the normal stress in these control elements exceeds the matrix cracking stress (300 MPa) these control elements will be released by reducing the normal stiffness of these elements to zero, thus, simulating the occurrence of the matrix cracking.
cracking. In the other 80 axi-symmetric interface elements used to model the fibre/matrix interface the shear stiffness \( K_s \) is set to a high value initially to limit any non-slip displacement. Once the interface shear stress exceeds the interface shear strength (100 MPa), the shear stiffness \( K_s \) is set to zero allowing slip. Loads were applied incrementally and stress-strain relationships have been obtained. Mesh convergence for this mesh has been checked by doubling the number of elements. The displacements of selected nodes has been found to vary only in the range of 2.5-5.5%. This indicates that the chosen FE mesh is a stable configuration.

Figure 4.5 presents the comparison of the axi-symmetric finite element simulated stress-strain response with the experimental responses obtained from testing seven specimen of the SiC/SiC composite and also with the response obtained from plane strain analysis of Rao & Mukherjee (1995). From this figure it can be observed that the axi-symmetric finite element simulated stress-strain response is with in the range of the experimentally obtained responses. It can be seen that the finite element simulated response traces almost a mean path of the responses obtained from testing seven specimens. It can be further observed that the stress-strain response obtained from the present investigation shows stiffer behaviour when compared to plane strain analysis of Rao and Mukherjee (1995). Also, it can be observed that the axisymmetric plane strain analysis under estimated the tensile strength of the Composite. This is expected because, in the axi-symmetric simulation, the realistic concentric cylindrical geometry of the fibre and matrix has been realised. On the other hand the plane strain analysis, assumes the matrix and fibre as layers of uniform thickness. By observing these results, it can be concluded that the present axi-symmetric finite element model is quite efficient and is suitable for the analysis of BMCs.
Fig. 4.5: Comparison of experimental and finite element simulated stress-strain responses for axisymmetric approximation.

4.5 CLOSURE

The development of the micromechanical axi-symmetric finite element model for analysing brittle-matrix-composites has been presented in detail. The model uses isoparametric axi-symmetric interface elements formulated in the previous chapter, for idealising the bi-material interface and hence model the interfacial mechanics realistically. This model alleviates the lacunae of other discrete spring element approaches for modelling the bi-material interface. The model has been extensively validated against the experimental results and plane strain solutions available in the literature. The application of the finite element model for conducting the parametric studies on BMCs to derive the optimum property profiles will be presented in the next chapter.