Development and Implementation of Computer Program

4.1 MATLAB Software:

MATLAB is an interactive system for doing numerical computations. The program got its name from the fact that it is a "Matrix Laboratory" because it is based on matrix manipulations, which produce high efficiency in calculations. Each entry into the MATLAB is treated as a matrix, even if it is a single number. Computation speed of the program is incredible so it is very suitable to be used to solve the large number of simultaneous equations resulting from finite element analyses. For example, the inverse of a very large stiffness matrix is calculated using one command in contrast to the old programming languages where that task usually requires a large amount of programming effort. [5]

4.2 Implementation of the FEM Analysis:

Special purpose of computer program was developed using MATLAB to calculate displacements and reactions at the nodes, displacement increments, stress increments, shear forces, bending moments, nodal residual forces and tangent stiffness matrix for the cases of circular and rectangular elements within each problem domain.

4.3 Outline of the Major Program Steps:

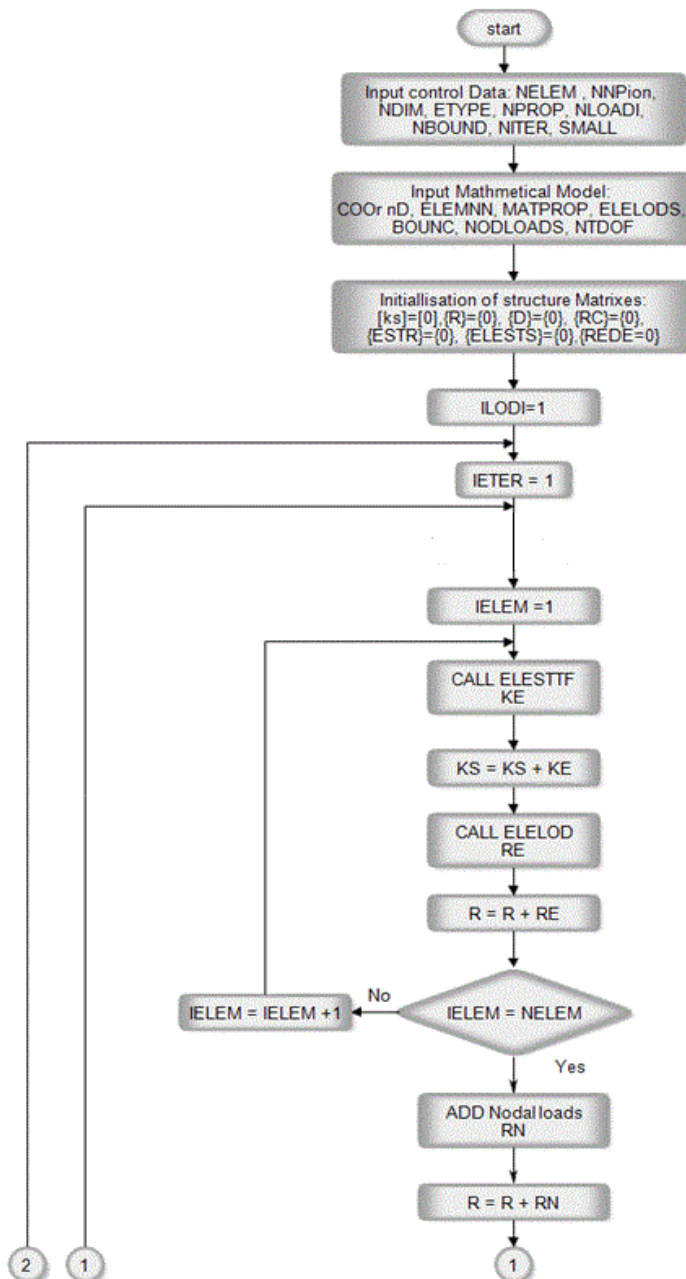The following is a summary of the basic stages that one has to go through when implementing the MATLAB developed computer program.
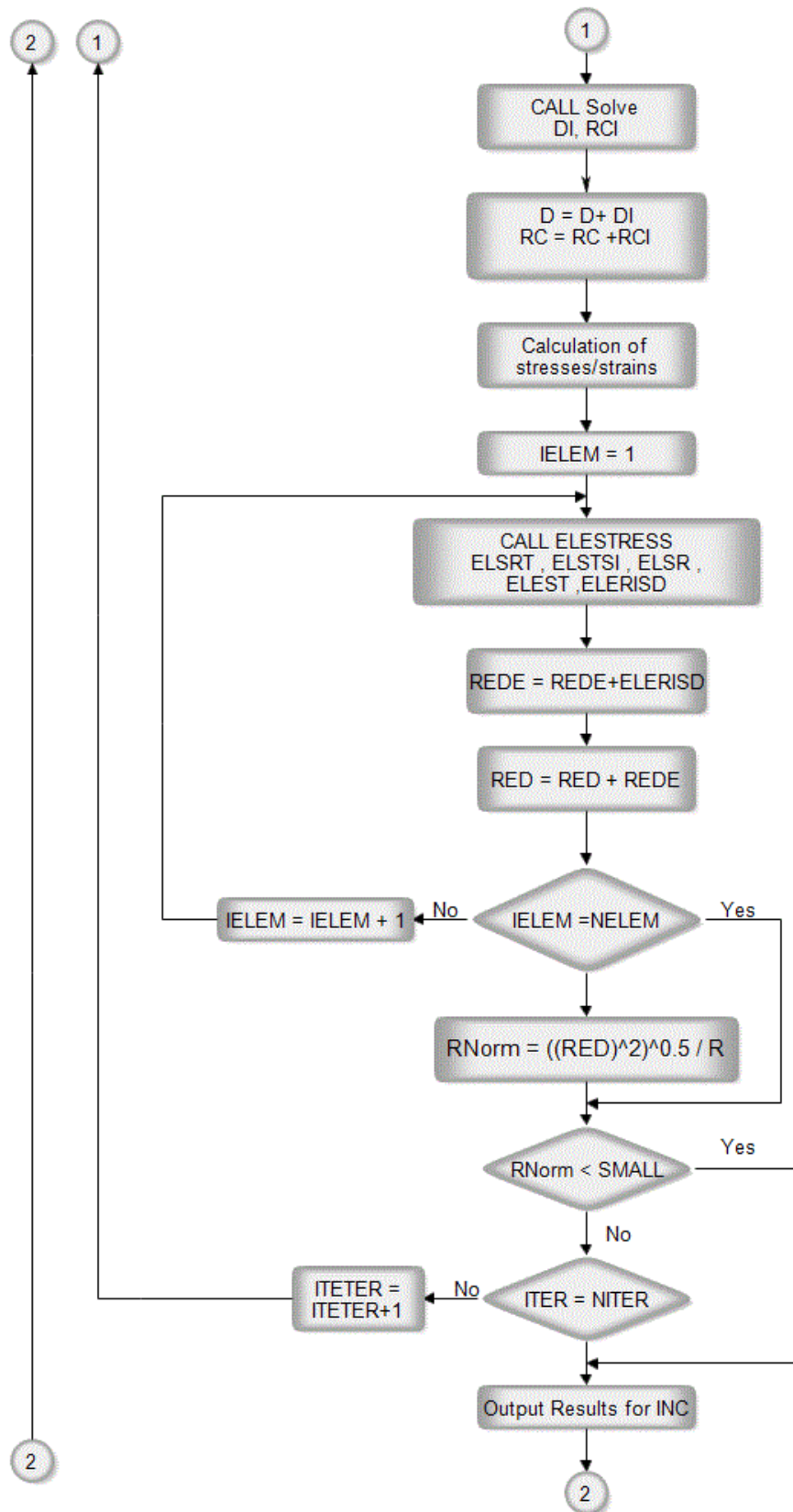
✓ Start.

✓ Ask for input data :

➤ Young's Modulus of beam.

➤ Cross section of beam, whether circular or rectangular and calculate I (moment of area). If the moment is available enter the value directly.

➢ Input the length of beam, number of elements wish to divide the beam into, length of each element.

✓ Calculate the element stiffness matrix and assemble that into global stiffness matrix.

✓ Input if there is a support at a node other than the fixed one for the cantilever.

✓ If there is a support, remove the row and column corresponding to the vertical displacement and store the reduced stiffness matrix as the displacement is constrained in **y** direction but angular displacement is allowed.

✓ After that remove the first, second and third row and column of the reduced global stiffness matrix as the angular and vertical displacements are constrained on the first node as there is a cantilever support.

✓ Input the forces position and value.

✓ After obtaining the reduced stiffness matrix and the force matrix, multiply the inverse of the reduced stiffness matrix and force matrix to obtain the nodal displacements.

✓ Calculate the:

o Reactions $\{Fr\}$.

o Displacement increments $\{a\}^{i+1} = \{a\}^i + \{\Delta a\}^i$.

o Stress increments $\{S^*\}^{i+1} = [D]\{\Delta\varepsilon^*\}^i$.

o Total stresses nodal residual forces $\{S^*\}^{i+1} = \{S^*\}^i + \{\Delta S^*\}^i$.

o Initial stress stiffness matrix $[K_\sigma] = \int [G]^T [P_i][G]dl$.

o Tangent stiffness matrix $[K_T] = [K_0] + [K_\sigma]$.

✓ Plot the displacement increments with length.

✓ Plot the stress increments with length.

✓ End.

4.4 Program Flow Chart

Figure (4.1) shows the usual flow chart of processes involved in the implementation of the finite element analysis. The flow chart for the computation of the element tangent stiffness matrix is shown in Figure (4.2). Figure (4.3) outlines the flow chart for calculation of element incremental strains/stress resultants and residuals. Appendix (C) contains the listing of the program.

```
                          ( start )
                             │
          ┌──────────────────────────────────────┐
          │ Input control Data: NELEM , NNPion,   │
          │   NDIM, ETYPE, NPROP, NLOADI,         │
          │       NBOUND, NITER, SMALL            │
          └──────────────────────────────────────┘
                             │
          ┌──────────────────────────────────────┐
          │       Input Mathmetical Model:        │
          │ COOr nD, ELEMNN, MATPROP, ELELODS,    │
          │      BOUNC, NODLOADS, NTDOF           │
          └──────────────────────────────────────┘
                             │
          ┌──────────────────────────────────────┐
          │ Initiallisation of structure Matrixes:│
          │  [ks]=[0],{R}={0}, {D}={0}, {RC}={0}, │
          │ {ESTR}={0}, {ELESTS}={0},{REDE=0}     │
          └──────────────────────────────────────┘
                             │
                        ( ILODI=1 )
                             │
                       ( IETER = 1 )
                             │
                       ( IELEM =1 )
                             │
                     ┌──────────────┐
                     │ CALL ELESTTF │
                     │      KE      │
                     └──────────────┘
                             │
                      ( KS = KS + KE )
                             │
                     ┌──────────────┐
                     │ CALL ELELOD  │
                     │      RE      │
                     └──────────────┘
                             │
                      ( R = R + RE )
                             │
            No           ◇ IELEM = NELEM ◇
( IELEM = IELEM +1 ) ◄───
                             │ Yes
                     ┌──────────────┐
                     │ ADD Nodal loads│
                     │      RN       │
                     └──────────────┘
                             │
                      ( R = R + RN )
                             │
  (2)  (1)               (1)
```

(2) (1)   (1)

                                    CALL Solve
                                    DI, RCI

                                    D = D+ DI
                                    RC = RC +RCI

                                    Calculation of
                                    stresses/strains

                                    IELEM = 1

                                    CALL ELESTRESS
                                    ELSRT , ELSTSI , ELSR ,
                                    ELEST ,ELERISD

                                    REDE = REDE+ELERISD

                                    RED = RED + REDE

    IELEM = IELEM + 1 ◄─No─ IELEM =NELEM ─Yes─►

                                    RNorm = ((RED)^2)^0.5 / R

                                                        Yes
                                    RNorm < SMALL ─────►

                                                        No

    ITETER = ◄─No─ ITER = NITER
    ITETER+1

                                    Output Results for INC

(2)
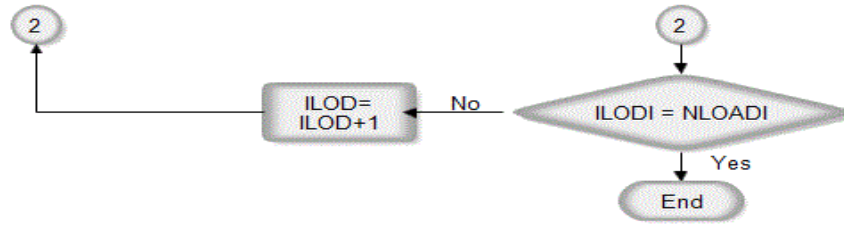
                                      (2)

Figure 4.1: Main Program Flow Chart
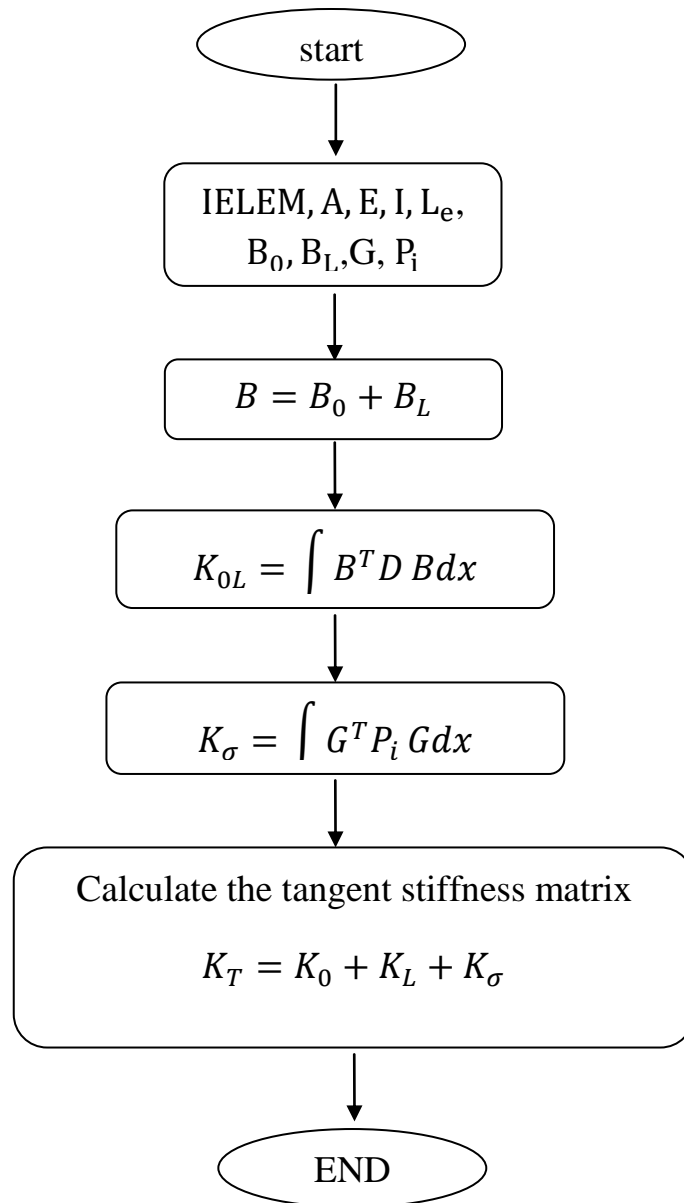
Figure 4.2: Flow chart to calculate tangent stiffness matrix

```
                    ╭─────────────╮
                    │    Start    │
                    ╰─────────────╯
                           │
                           ▼
       ╭──────────────────────────────────────╮
       │ IELEM, A, E, I, Lₑ,B₀, Bₗ,ds,dEs,σ    │
       │           , Resid,D                    │
       ╰──────────────────────────────────────╯
                           │
                           ▼
            ╭──────────────────────────╮
            │      B = B₀ + Bₗ          │
            ╰──────────────────────────╯
                           │
                           ▼
    ╭────────────────────────────────────────────────╮
    │ Δε = (B₀ + Bₗ(dEs) + ½Bₗ(dEs)) ds              │
    ╰────────────────────────────────────────────────╯
                           │
                           ▼
            ╭──────────────────────────╮
            │        Δσ = D Δε          │
            ╰──────────────────────────╯
                           │
                           ▼
            ╭──────────────────────────╮
            │        σ = σ + Δσ         │
            ╰──────────────────────────╯
                           │
                           ▼
            ╭──────────────────────────╮
            │     RINT(6, IELEM)        │
            ╰──────────────────────────╯
                           │
                           ▼
            ╭──────────────────────────╮
            │          READ             │
            ╰──────────────────────────╯
                           │
                           ▼
                    ╭─────────────╮
                    │     END     │
                    ╰─────────────╯
```

The boxes in the flow chart read:

Start

IELEM, A, E, I, L_e, B_0, B_L, ds, dEs, σ, Resid, D

$B = B_0 + B_L$

$\Delta\varepsilon = (B_0 + B_L(dEs) + \frac{1}{2}B_L(dEs))\ \mathrm{ds}$

$\Delta\sigma = D\ \Delta\varepsilon$

$\sigma = \sigma + \Delta\sigma$

RINT(6, IELEM)
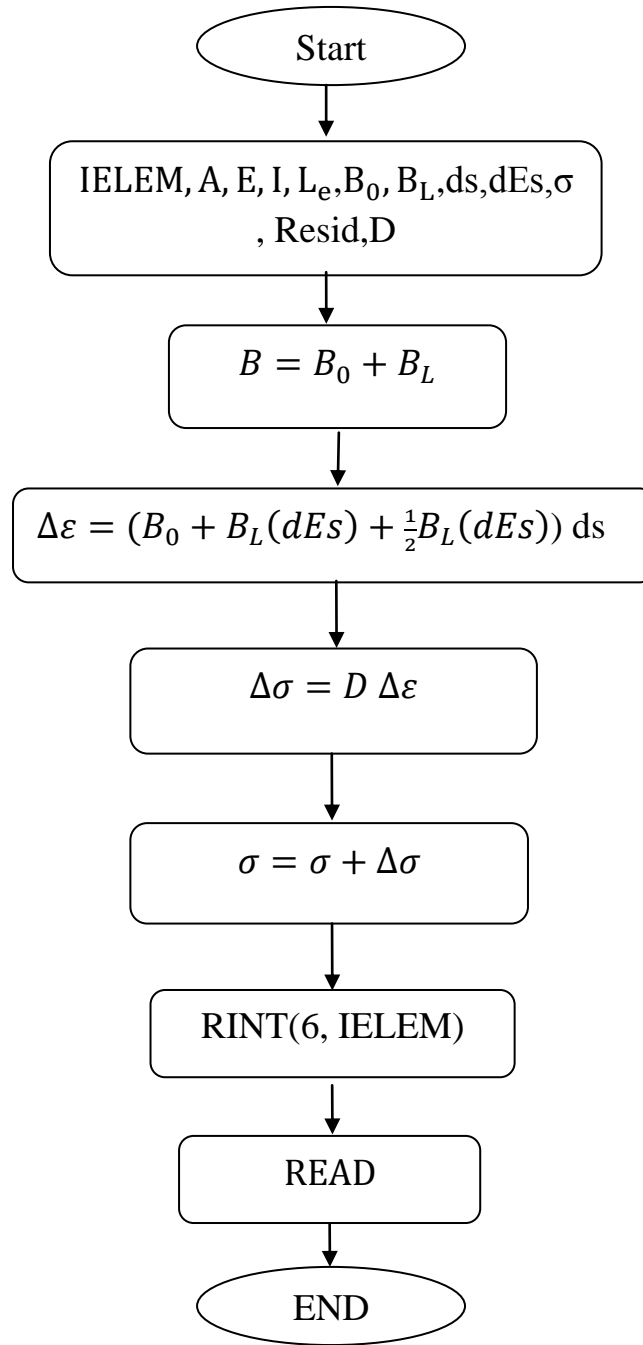
READ

END

Figure 4.3: Flow chart to calculate Strain/Stress resultants and residuals

Table 4.1: Required Input Data Read by Program

Data	Description
NELEM	Number of Element
NNPoin	Number of Point
NDIM	Number of Dimensions
ETYPE	Element Type
NPROP	Number of Properties Material
NLOADI	Number of Load Increments
NBOUND	Number of Boundary Conditions
NITER	Number of Iteration
SMALL	Percentage to clearance to terminate iterations
COOr nD	Coordinates of Points
MATPROP	Materials Properties
ELELODS	Element Loads
BOUNDC	Boundary Conditions
NODLOADS	Loads in Nodes
NTDOF	Number of Degrees of freedom
ESTR	Element Strains
ELESTS	Element Stresses
RINT	Internal forces
REDE	Residual