This script implements a numerical analysis simulation using the finite element method (FEM) for stress analysis. It calculates displacements, reactions and stress. The deformed mesh is plotted and the stress contour map.

The script is designed to handle a mesh composed of quadrilateral elements (2D). The mesh is created in Femap, and the data related to elements and nodes is exported to Python via .csv files.

Eight CSV files are required:

- Four input files (element_input.csv, entry_nodes.csv,entry_restrictions.csv and entry_loads.csv) containing data from Femap.

- Four blank output files (element_output.csv, output_nodes.csv, output_restrictions.csv and output_loads.csv), which will be populated after running the script.

Explanation of the finite element method for structural analysis.

Summary of Steps

1. **Initialize element properties** (nodes, material constants, thickness).
2. **Define shape functions and their derivatives** to compute the Jacobian.
3. **Compute B matrix** to relate nodal displacements to strains.
4. **Define the material matrix D** for plane stress or plane strain.
5. **Assemble the element stiffness matrix K** using Gaussian quadrature.
6. **Solve the system** $\mathbf{K}\,\mathbf{U} = \mathbf{F}$ for nodal displacements.
7. **Compute stress** $\boldsymbol{\sigma} = \mathbf{D}\,\mathbf{B}\,\mathbf{U}$ at desired locations.

This is a standard approach in FEM for 2D elasticity problems using bilinear quadrilateral elements.

Problem Statement

We consider a 2D elasticity problem where a quadrilateral element is subjected to external forces. The goal is to compute nodal displacements and stresses using FEM.

Assumptions:

- We use bilinear 4-node quadrilateral elements.
- The material follows linear elasticity.
- We solve in plane stress or plane strain conditions.
- The problem is discretized into multiple elements, and the local stiffness matrices are assembled into a global stiffness matrix.

Step 1: Define the Geometry and Nodes

We define a quadrilateral element with 4 nodes.

Each node has two degrees of freedom (DOF): displacement in x and y directions.

Total DOFs per element = 8 (since 4 nodes × 2 DOFs per node).

Step 2: Shape Functions and Natural Coordinates

In the **natural coordinate system** (s, t ranging from -1 to 1), the shape functions $N_i(s, t)$ are:

$$N_1 = \frac{1}{4}(1-s)(1-t), \quad N_2 = \frac{1}{4}(1+s)(1-t)$$

$$N_3 = \frac{1}{4}(1+s)(1+t), \quad N_4 = \frac{1}{4}(1-s)(1+t)$$

These functions interpolate the displacements within the element.

Step 3: Compute the Jacobian Matrix

The **Jacobian matrix** J transforms derivatives from local (s, t) coordinates to global (x, y):

$$J = \begin{bmatrix} \frac{\partial x}{\partial s} & \frac{\partial y}{\partial s} \\ \frac{\partial x}{\partial t} & \frac{\partial y}{\partial t} \end{bmatrix}$$

Each term is computed as:

$$\frac{\partial x}{\partial s} = \sum_{i=1}^{4} \frac{\partial N_i}{\partial s} x_i, \quad \frac{\partial y}{\partial s} = \sum_{i=1}^{4} \frac{\partial N_i}{\partial s} y_i$$

$$\frac{\partial x}{\partial t} = \sum_{i=1}^{4} \frac{\partial N_i}{\partial t} x_i, \quad \frac{\partial y}{\partial t} = \sum_{i=1}^{4} \frac{\partial N_i}{\partial t} y_i$$

The determinant of the Jacobian gives the area scaling factor.

Step 4: Compute the Strain-Displacement Matrix B

The strain-displacement matrix B relates nodal displacements to strains:

$$B = \begin{bmatrix} \frac{\partial N_1}{\partial x} & 0 & \frac{\partial N_2}{\partial x} & 0 & \frac{\partial N_3}{\partial x} & 0 & \frac{\partial N_4}{\partial x} & 0 \\ 0 & \frac{\partial N_1}{\partial y} & 0 & \frac{\partial N_2}{\partial y} & 0 & \frac{\partial N_3}{\partial y} & 0 & \frac{\partial N_4}{\partial y} \\ \frac{\partial N_1}{\partial y} & \frac{\partial N_1}{\partial x} & \frac{\partial N_2}{\partial y} & \frac{\partial N_2}{\partial x} & \frac{\partial N_3}{\partial y} & \frac{\partial N_3}{\partial x} & \frac{\partial N_4}{\partial y} & \frac{\partial N_4}{\partial x} \end{bmatrix}$$

Step 5: Compute the Constitutive Matrix D

For plane stress:

$$D = \frac{E}{1 - \nu^2} \begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \frac{1-\nu}{2} \end{bmatrix}$$

For plane strain:

$$D = \frac{E}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1-\nu & \nu & 0 \\ \nu & 1-\nu & 0 \\ 0 & 0 & \frac{1-2\nu}{2} \end{bmatrix}$$

Step 6: Compute the Stiffness Matrix Ke

The element stiffness matrix is:

$$K_e = \int_A B^T D B \, t \, dA$$

We use Gaussian quadrature with 4 points:

$$K_e = \sum_{i=1}^{4} B_i^T D B_i \cdot t \cdot |J| \cdot W_i$$

Step 7: Assemble the Global Stiffness Matrix K

Each element contributes to the global stiffness matrix K:

$$K = \begin{bmatrix} K_{11} & K_{12} & K_{13} & K_{14} \\ K_{21} & K_{22} & K_{23} & K_{24} \\ K_{31} & K_{32} & K_{33} & K_{34} \\ K_{41} & K_{42} & K_{43} & K_{44} \end{bmatrix}$$

Each Ke is mapped to the correct global positions using node indexing.

Step 8: Apply Boundary Conditions

- Some nodes are fixed → Set displacement values to zero.
- Modify K and the force vector F accordingly.

Step 9: Solve for Displacements

$$KU = F$$

Solve for U using Gaussian elimination or a solver.

Step 10: Compute Stresses

Once U is known:

$$\sigma = DBU$$

This gives stresses at the Gauss points or centroid.