Development of 6-Node Hexagon shaped plane stress element and its static behaviour analysis.

J.R.Mevada and Kaushal Bhavsar

1Associate Prof., Mechanical Engg. Dept., Ganpat University, Kherva, Mehsana, Gujarat 382711, India
E-mail: jignesh.mevada@ganpatuniversity.com
2Student (M. Tech. CAD/ CAM), Mechanical Engg. Dept., Ganpat University, Kherva, Mehsana, Gujarat 382711, India
E-mail: kaushalbhavsar87@gmail.com

Abstract:
A new element is introduced for FEA family in this paper. Different geometries have different response in physical world when implemented in FEA solver. Here development of new hexagon shaped element is introduced. Hexagon geometry can work well for isotropic materials. Hexagon shaped element can give us response for 6 vertices and 6 edges. A hexagon shaped element is more effective as compared to tria or quard element for isotropic material. To check this shape functions are formulated from geometry. A code is generated in MATLAB for plane stress response of 6-Node Hexagon element. This code contains stiffness matrix formulation, deflection formulation. The developed element responds with change in thickness. The performance of new element has been compared with analytical results.

The behaviour from given element is same as behaviour of any beam structure or not is checked in same physics and it also contains same formulation which is required in FEA solver.

KEY WORDS: 6 node Hexagon element, hexagon finite Element, Plane stress element in FEA, Hexagon element.

Introduction:
In FEA world we have few definite elements like for 1-D we have line element and as well for 2-D we are having 2-D spar element, Beam element, Pipe element, Tria element(3 node, 6 node, etc.), Quad element (4 node, 8 node, etc...) and for 3-D we have tetrahedral as well hexahedral element.

Geometrical continuity of hexagon shaped element is $C_0$. “If two adjacent elements are generated from parent, in which the shape continuity satisfy $C_0$ continuity requirements then the distorted elements will be continuous and compatible.”[12]

The basic formulation of FEA element requires shape function. From geometrical criteria shape functions are generated. The element is defined in $\xi$ and $\eta$ parametric co-ordinates and it contains hexagon shape with following sub forms.

$$\xi = \frac{2x}{a} ; \eta = \frac{y}{a \sin \theta}$$

(1)

Here hexagon shaped plane stress element with thickness is defined and mathematical model is generated for it. Actual representation of element is like below it contains 6 node 6 vertices and also thickness, it gives same presentation as solid element in plane stress with thickness. So element can be represented as 6 node hexagon element for plane stress with thickness formulation.
Developed Langrangian shape functions for hexagonal shaped element at different node,

\[ N_1 = \frac{1}{8} (1 - \xi) (1 - \eta) \]  
\[ N_2 = \frac{1}{8} (1 + \xi) (1 - \eta) \]  
\[ N_3 = \frac{1}{8} (2 - 2\eta + \xi) \]  
\[ N_4 = \frac{1}{8} (1 + \xi) (1 + \eta) \]  
\[ N_5 = \frac{1}{8} (1 - \xi) (1 + \eta) \]  
\[ N_6 = \frac{1}{8} (1 + 2\eta - \xi) \]  

\[ \sum_{i=1}^{6} N_i = 1 \]

Shape functions satisfy the condition that summation of shape function must be unity.

Material properties matrix is generated for Iso-tropic material. The material has same elastic properties in all direction. Here the material properties are accepted for steel. \[11\]

\[ D_{3\times3} = \begin{bmatrix} E' & E'\nu & 0 \\ E'\nu & E' & 0 \\ 0 & 0 & G \end{bmatrix} \]

Where

\[ E' = \text{Young’s modulus} = E/(1 - \nu^2) \]
\[ G = \text{Shear modulus of rigidity} = E / 2(1 + \nu) \]
\[ \nu = \text{Poisson’s ratio} \]

Take any of the material and consider isotropic property.

**JACOBIAN MATRIX FORMULATION**\[11, 12, 13\]

\[ J = \begin{bmatrix} J_{11} & J_{12} \\ J_{21} & J_{22} \end{bmatrix} \]

Where

\[ J_{11} = \frac{\partial x}{\partial \xi} ; J_{12} = \frac{\partial y}{\partial \xi} \]
\[ J_{21} = \frac{\partial x}{\partial \eta} ; J_{11} = \frac{\partial y}{\partial \eta} \]
\[ \det J = \partial [J_{22} \times J_{11} - J_{21} \times J_{12}] \]
We have Jacobian components by using them we can easily find A matrix as well G matrix for finding strain as well displacement matrix.[12]

\[ x = (N_1x_1 + N_2x_2 + N_3x_3 + N_4x_4 + N_5x_5 + N_6x_6) \]
\[ y = (N_1y_1 + N_2y_2 + N_3y_3 + N_4y_4 + N_5y_5 + N_6y_6) \]

where

\[ N_1, N_2, ..., N_6 = \text{shape functions for node}1\text{to}6. \]
\[ x_1, x_2, ..., x_6 = \text{co ordinates of}1\text{st element in} x. \]
\[ y_1, y_2, ..., y_6 = \text{co ordinates of}1\text{st element in} y. \]

\[ A = \frac{1}{\det J} \begin{bmatrix}
  J_{22} & -J_{12} & 0 & 0 \\
  0 & 0 & -J_{21} & J_{11} \\
  -J_{21} & J_{11} & J_{22} & -J_{12}
\end{bmatrix} \]  

\[ G = \begin{bmatrix}
  \frac{\partial N_1}{\partial \xi} & 0 & \ldots & 0 & \frac{\partial N_6}{\partial \xi} & 0 \\
  \frac{\partial N_1}{\partial \eta} & 0 & \ldots & 0 & \frac{\partial N_6}{\partial \eta} & 0 \\
  0 & \frac{\partial N_1}{\partial \xi} & 0 & \ldots & \frac{\partial N_5}{\partial \xi} & 0 & \frac{\partial N_6}{\partial \xi} \\
  0 & \frac{\partial N_1}{\partial \eta} & 0 & \ldots & \frac{\partial N_5}{\partial \eta} & 0 & \frac{\partial N_6}{\partial \eta}
\end{bmatrix} \]

\[ B = A \times G \]

**STIFFNESS MATRIX (K_e) GENERATION:**

The numerical integration technique associated with the parametric co-ordinate system \( \xi \) and \( \eta \) related to the element matrices is Gauss Legendry Quadrature method. The sampling points defined are 2. [12,13]

Integrating strain matrix formulation with \( \xi \) and \( \eta \) like below, we get stiffness matrix of size 12×12.

\[ K^e = t_e \int_{-1}^{1} \int_{-1}^{1} B^T D B (\det J) \, d\xi \, d\eta \]

\[ \phi(\xi, \eta) = t_e \int_{-1}^{1} (B^T \times D \times B \times \det J) \, d\eta \]

\[ K_{ij} = [(\omega_1)^2 \times \phi(\xi_1, \eta_1)] + [\omega_1 \times \omega_2 \times \phi(\xi_1, \eta_2)] + [(\omega_2 \times \omega_1) \times \phi(\xi_2, \eta_1)] + [(\omega_2)^2 \times \phi(\xi_2, \eta_2)] \]

For, 2 sampling point application we do have weights like below. As sampling points increases the weights are changing. For said condition the weight is 1. And parametric co-ordinates do have said values. [12]

\[ \omega_1 = \omega_2 = 1 \]
\[ \xi_1 = \eta_1 = -0.5773502692 \]
\[ \xi_2 = \eta_2 = 0.5773502692 \]

Stiffness matrix developed by this method will have the formation of 12 × 12 matrix. To solve a single element problem we do have the formulation like
For single element generation we are getting resultant matrix that is in form of deflection matrix which is $Q_{12 \times 1}$.

Now testing the element response in physical condition we can get different results for constant thickness and variable load as well constant load and variable thickness conditions. To check performance of this element two physical conditions are created. Here case 1 contains a straight beam with constant thickness and case 2 contains taper beam with constant thickness.

**CASE :1** Testing of element for axial loading condition for simple beam:

![Figure 2: Beam with 3 element mesh](image)

**CASE :2** Testing of element for axial loading condition for taper Beam:

![Figure 3: Taper beam with 3 element mesh](image)

The physical condition is meshed by 3 hexagon shaped elements. As each node has 2 D.O.F. we have 12 D.O.F. for each element. Node 3, 4, 5 and 7 are common node in element 1, element 2 and element 3 as shown in figure. So we can find 13 nodes in first mesh. Here 13 nodes have been generated and each node contains 2 D.O.F. So the meshed structure does have 26 D.O.F. The equation generated has following formulation. Material properties have been taken for mild steel.

$$[K_{26 \times 26}] \times [Q_{26 \times 1}] = [F_{26 \times 1}]$$

(21)

Load is applied to node 12. And assumption made that node 6 and 10 are rigid node with zero deflection and zero force. Implementing gauss elimination approach for defining boundary conditions we can get deformation values for defined physical condition. After eliminating 2 node stiffness and deflection values we have following formulation.
\[ \begin{bmatrix} K_{22 \times 22} \end{bmatrix} \times \begin{bmatrix} Q_{22 \times 1} \end{bmatrix} = \begin{bmatrix} F_{22 \times 1} \end{bmatrix} \]  
(22)

From the implementation of hexagon shaped element we can get results like below.

Result and Discussion:
Here analytical and Matlab code results have been compared and the comparison is like below. With same geometry we can get comparative results for higher thickness value of the developed hexagon shaped solid element. From this element development we can get the similar behaviour as a system has for the defined physical condition what we get from analytical results.

![Displacement Vs Normalised length chart from code results.](image1)

**CASE: 1**

![Deflection Vs. Load Diagram. THICKNESS = 1 mm.](image2)

Figure 5 shows comparison between behaviour of 3 element mesh code for developed element with analytical results for physical condition variable load on straight beam with constant thickness 1 mm.
Figure 6 shows comparison between behaviour of 3 hexagonal element mesh code for developed element with analytical results for physical condition variable load on straight beam with constant thickness 2mm.

Figure 7 shows comparison between behaviour of hexagon 3 element mesh code with analytical results for physical condition variable load on straight beam with constant thickness 3mm.

CASE :2

Figure 8 shows comparison between behaviour of hexagon shaped 3 element mesh code with analytical results for physical condition variable load on taper beam with constant thickness 1mm.
CASE: 1
The results from this element from a straight beam case are comparative good and the error established is 5.05% for 10,000 N loading conditions at 1 mm thickness. As thickness increases to 2 mm error is 2.813% for 10,000 N. For 3 mm thickness error is nearer to 1.90% for 10,000 N loading. As thickness increases the error becomes low. The developed code and developed element implementation is possible to analyse different physical conditions.

CASE: 2
The results from this element from a taper bar case; the error established is 8.28% for 10,000 N loading conditions at 1 mm thickness. As thickness increases to 2 mm error is 4.13% for 10,000 N. For 3 mm thickness error is nearer to 2.80% for 10,000 N loading. As thickness increases the error becomes low. The developed code and developed element implementation is possible to analyse different physical conditions.

The errors do have specific formation so it can be eliminated by implementation of additional error rectification terms. Introduced 6-Node Hexagon Shaped Element is effective with implementation of some numerical error removal code numerical error removal is possible

Conclusion:
Hexagon shaped element developed for plane stress with thickness formulation is done effectively. The code developed in MATLAB gives comparative same plots which are actually possible by analytical...
approach. The model proposed in this paper gives us the same result which an analytical approach gives. So it is effective and implementable to analyse different physical conditions of plane stress.

References: