An Abaqus user element (UEL) implementation of linear elastostatics

Bibekananda Datta

Mechanical Engineering, Johns Hopkins University, Baltimore, MD 21211, USA

The contents presented in this document are adapted from standard finite element literature (Fish & Belytschko, 2007; Hughes, 1987; Zienkiewicz et al., 2013, 2014). Interested users are suggested to consult textbooks on finite element modeling for solid mechanics, tutorials and textbooks on Fortran programming, and Abaqus documentation (Dassault Systèmes, 2024) for details.

1 Summary of the finite element formulation

The governing partial differential equation for stress equilibrium as well as the boundary conditions in terms of the displacement vector, $\mathbf{g}$, on Γ_g and the Cauchy traction, $\mathbf{t}$, on Γ_t are given by,

$$\begin{aligned} \operatorname{div}(\boldsymbol{\sigma}) + \rho\mathbf{b} &= 0 \qquad &&\text{in } \Omega, \\ \mathbf{u} &= \mathbf{g} &&\text{on } \Gamma_g, \\ \boldsymbol{\sigma}\cdot\mathbf{n} &= \mathbf{t} &&\text{on } \Gamma_t, \end{aligned} \tag{1.1}$$

where, $\mathbf{b}$ is the body force per mass and ρ is the density of the material.

The global weak form, $\mathcal{W}_{\mathbf{u}}(\mathbf{u})$, can be written as,

$$\mathcal{W}_{\mathbf{u}}(\mathbf{u}) = -\int_{\Omega} \boldsymbol{\sigma}\left(\boldsymbol{\varepsilon}(\mathbf{u})\right) : \operatorname{grad}(\mathbf{w})\ dv + \int_{\Omega} \rho\mathbf{b}\cdot\mathbf{w}\ dv + \int_{\Gamma_t} \mathbf{t}\cdot\mathbf{w}\ ds = 0, \tag{1.2}$$

where, $\mathbf{u} \in \mathscr{U}$ be the trial solution which satisfies $u_i = g_i$, and $\mathbf{w} \in \mathscr{W}$ be a vector test (or weight) function which satisfies $w_i = 0$ on Γ_g, and for small deformation, the infinitesimal strain tensor can be defined as,

$$\boldsymbol{\varepsilon} = \frac{1}{2}\left[\operatorname{grad}(\mathbf{u}) + (\operatorname{grad}(\mathbf{u}))^\top\right] = \operatorname{sym}[\operatorname{grad}(\mathbf{u})], \tag{1.3}$$

and the constitutive relation for isotropic linear elastic material is given by,

$$\boldsymbol{\sigma} = \mathbb{C} : \boldsymbol{\varepsilon}, \qquad \Rightarrow \boldsymbol{\sigma} = \lambda\,\operatorname{tr}(\boldsymbol{\varepsilon})\mathbb{1} + 2\mu\boldsymbol{\varepsilon}, \qquad \Rightarrow \sigma_{ij} = \mathbb{C}_{ijkl}\varepsilon_{kl} = \lambda\varepsilon_{kk}\delta_{ij} + 2\mu\varepsilon_{ij}, \tag{1.4}$$

where, $\boldsymbol{\sigma}$ is known as the Cauchy stress. Here, $\mathbb{c}$ is known as the fourth-order elasticity or modulus tensor, and μ and λ are known as Lamé constants. In finite element implementation, the constitutive relations are reduced to a matrix-vector form using the Voigt notation convention.

Using the standard Galerkin discretization, the discretized residual for each element, $\mathbf{R}^e_{\mathbf{u}}(\mathbf{u}_e)$, can be written as,

$$\mathbf{R}^e_{\mathbf{u}}(\mathbf{u}_e) = -\int_{\Omega^e} \mathbf{B}_{\mathbf{u}}^\top \boldsymbol{\sigma}\left(\boldsymbol{\varepsilon}(\mathbf{u}_e)\right)\ dv + \int_{\Omega^e} \rho \mathbf{N}_{\mathbf{u}}^\top \mathbf{b}^e\ dv + \int_{\Gamma^e_t} \mathbf{N}_{\mathbf{u}}^\top \mathbf{t}^e\ ds = 0, \tag{1.5}$$

where, $[\mathbf{N}^a_{\mathbf{u}}]$ is the nodal matrix and $[\mathbf{N}_{\mathbf{u}}]$ is the element matrix of the interpolation functions, and for a three-dimensional case, they are given by,

$$\mathbf{N}^a_{\mathbf{u}} = \begin{bmatrix} N_i & 0 & 0 \\ 0 & N_i & 0 \\ 0 & 0 & N_i \end{bmatrix},\ \mathbf{N}_{\mathbf{u}} = \begin{bmatrix} N_1 & 0 & 0 & N_2 & 0 & 0 & \cdots & N_{\text{nnode}} & 0 & 0 \\ 0 & N_1 & 0 & 0 & N_2 & 0 & \cdots & 0 & N_{\text{nnode}} & 0 \\ 0 & 0 & N_1 & 0 & 0 & N_2 & \cdots & 0 & 0 & N_{\text{nnode}} \end{bmatrix}, \tag{1.6}$$

and the strain-displacement matrix, *i.e.*, the symmetric gradient matrix at the element level, $[\mathbf{B}_{\mathbf{u}}]$, and at the node level, $[\mathbf{B}^a_{\mathbf{u}}]$, are given by,

$$\mathbf{B}_{\mathbf{u}} = \begin{bmatrix} \mathbf{B}^1_{\mathbf{u}} & \mathbf{B}^2_{\mathbf{u}} & \mathbf{B}^3_{\mathbf{u}} & \cdots & \cdots & \mathbf{B}^{\text{nnode}}_{\mathbf{u}} \end{bmatrix}, \quad \text{and} \quad \mathbf{B}^a_{\mathbf{u}} = \begin{bmatrix} N^a_{,1} & 0 & 0 \\ 0 & N^a_{,2} & 0 \\ 0 & 0 & N^a_{,3} \\ 0 & N^a_{,3} & N^a_{,2} \\ N^a_{,3} & 0 & N^a_{,1} \\ N^a_{,2} & N^a_{,1} & 0 \end{bmatrix}. \tag{1.7}$$

Corresponding to the interpolation function matrix, $[\mathbf{N}_{\mathbf{u}}]$, and strain-displacement matrix, $[\mathbf{B}_{\mathbf{u}}]$, the degrees of freedom of an element, $\mathbf{u}_e$, and stress components in Voigt vector form, $\boldsymbol{\sigma}$, are represented in the following manner.

$$\begin{gathered} \mathbf{u}_e = \begin{bmatrix} u^1_1 & u^1_2 & u^1_3 & u^2_1 & u^2_2 & u^2_3 & \cdots & \cdots & \cdots & u^{\text{nnode}}_1 & u^{\text{nnode}}_2 & u^{\text{nnode}}_3 \end{bmatrix}^\top_{\text{ndim}*\text{nnode}\times 1}, \\ \text{and } \boldsymbol{\sigma} = \begin{bmatrix} \sigma_{11} & \sigma_{22} & \sigma_{33} & \sigma_{23} & \sigma_{13} & \sigma_{12} \end{bmatrix}^\top \end{gathered} \tag{1.8}$$

Now, the element stiffness matrix, $\mathbf{k}^e_{\mathbf{uu}}$, is given by,

$$\mathbf{k}^e_{\mathbf{uu}} = -\frac{\partial \mathbf{R}^e_{\mathbf{u}}}{\partial \mathbf{u}_e} = \int_{\Omega^e} \mathbf{B}_{\mathbf{u}}^\top \mathbf{D} \mathbf{B}_{\mathbf{u}}\ dv \tag{1.9}$$

where $\mathbf{D}$ represents the Voigt matrix form of the elasticity tensor which is also referred to as the stiffness matrix of the material in the context of the linear elasticity.

A simplified pseudo-code or procedure for implementing the above finite element model through the Abaqus `UEL(...)` subroutine is given below.

Procedure 1: Small strain UEL subroutine implementation in Abaqus/Standard

Input : `PROPS, COORDS, JELEM, JTYPE, NNODE, NDOFEL, TIME, DTIME, U, DU, V, A, PREDEF, JDLTYP, NDLOAD, MDLOAD, DDLMAG, ALMAG`

Output : `AMATRX, RHS, PNEWDT, ENERGY, SVARS`

Get $\mathtt{nInt} \leftarrow \mathtt{PROPS}$ and $\mathtt{ndim}, \mathtt{nstress} \leftarrow \mathtt{JTYPE}$
Initialize $\{\mathbf{N}^a_{\mathbf{u}}, \mathbf{B}^a_{\mathbf{u}}, \mathbf{N}_{\mathbf{u}}, \mathbf{B}_{\mathbf{u}}, \mathbf{k}^e_{\mathbf{uu}}\} = 0$
Get nodal displacement vector of the element, $\mathbf{u}_e, \Delta\mathbf{u}_e$
Get $w_{\text{int}}, \boldsymbol{\xi}_{\text{int}} \leftarrow$ SUBROUTINE $\mathtt{gaussQuadrtr}(\mathtt{ndim}, \mathtt{nNode})$
for $k = 1$ **to** nInt **do**
    Get $[\mathbf{N}_{\mathbf{u}}], \left[\frac{\partial \mathbf{N}_{\mathbf{u}}}{\partial \boldsymbol{\xi}}\right] \leftarrow$ SUBROUTINE $\mathtt{interpFunc}(\mathtt{ndim}, \mathtt{nNode})$
    Calculate $\frac{\partial \mathbf{x}}{\partial \boldsymbol{\xi}} = [\overline{\mathbf{x}}_e] \left[\frac{\partial \mathbf{N}_{\mathbf{u}}}{\partial \boldsymbol{\xi}}\right]$
    Calculate $\frac{\partial \mathbf{N}_{\mathbf{u}}}{\partial \mathbf{x}} = \frac{\partial \mathbf{N}_{\mathbf{u}}}{\partial \boldsymbol{\xi}} \left(\frac{\partial \mathbf{x}}{\partial \boldsymbol{\xi}}\right)^{-1}$ and J_ξ
    for $i = 1$ **to** nNode **do**
        Form $\mathbf{N}^a_{\mathbf{u}}(\mathtt{ndim}, \mathtt{NDOFEL})$ and $\mathbf{B}^a_{\mathbf{u}}(\mathtt{nstress}, \mathtt{ndim})$ matrix
        $\mathbf{N}_{\mathbf{u}}(1:\mathtt{ndim}, \mathtt{ndim} * (\mathtt{i}-1)+1 : \mathtt{ndim} * \mathtt{i}) = \mathbf{N}^a_{\mathbf{u}}(1:\mathtt{ndim}, 1:\mathtt{ndim})$
        $\mathbf{B}_{\mathbf{u}}(1:\mathtt{nstress}, \mathtt{ndim} * (\mathtt{i}-1)+1 : \mathtt{ndim} * \mathtt{i}) = \mathbf{B}^a_{\mathbf{u}}(1:\mathtt{nstress}, 1:\mathtt{ndim})$
    end
    `// end of nodal loop`
    Calculate $\boldsymbol{\varepsilon} = \mathbf{B}_{\mathbf{u}} \mathbf{u}_e$
    Get $\boldsymbol{\sigma}, \mathbf{D} \leftarrow$ SUBROUTINE $\mathtt{umatElastic}(\mathtt{PROPS}, \boldsymbol{\varepsilon})$ `// UMAT returns Cauchy stress and material tangent in Voigt form`
    $\mathbf{k}^e_{\mathbf{uu}} = \mathbf{k}^e_{\mathbf{uu}} + w_{\text{int}}(k) \det(J_\xi) \left(\mathbf{B}^\top_{\mathbf{u}} \mathbf{D} \mathbf{B}_{\mathbf{u}}\right)$
    $\mathbf{R}^e_{\mathbf{u}} = \mathbf{R}^e_{\mathbf{u}} - w_{\text{int}}(k) \det(J_\xi) \left(\mathbf{B}^\top_{\mathbf{u}} \boldsymbol{\sigma} - \rho \mathbf{N}^\top_{\mathbf{u}} \mathbf{b}\right)$
end
`// end of integration point loop`
Assign $\mathtt{AMATRX} = \mathbf{k}^e_{\mathbf{uu}}$ and $\mathtt{RHS} = \mathbf{R}^e_{\mathbf{u}}$

Remark 1. The current implementation includes three-dimensional elements, two-dimensional plane and axisymmetric elements with isotropic linear elastic material. For two-dimensional cases (plane and axisymmetric), the form and dimension of the interpolation function matrix, $[\mathbf{N}_{\mathbf{u}}]$, strain-displacement matrix, $[\mathbf{B}_{\mathbf{u}}]$, nodal degrees of freedom, $\mathbf{u}_e$, stress vector, $\boldsymbol{\sigma}$, and material tangent matrix $[\mathbf{D}]$ are needed to be modified. See Appendix A for brief discussion and Hughes, 1987; Zienkiewicz et al., 2013, 2014 for further details. Since Abaqus does not natively support viewing element-level output (stress and strain) from the user elements, a layer of additional dummy elements with Abaqus-provided user subroutine, `UVARM(...)`, was used to view the element-level results. Additionally, these same set of dummy elements can be used to apply body force and traction-type boundary conditions on the user elements. This is why, the current implementation does not include body force and traction boundary condition implementation.

2 Modeling in Abaqus

If and only if the user element has the same topology as any standard built-in element available in Abaqus, we can build the primary model in Abaqus/CAE, export the input file (`.inp`), and modify it to execute with Abaqus/Standard solver. Building the primary model is a standard exercise, and it is strongly recommended to check the option **Do not use parts and assemblies in input files** from the **model attributes** drop-down menu before exporting the input file of a model. Once the input is available, it must be modified following the instructions in the Abaqus user manual (Dassault Systèmes, 2024).

The first step is to include the element definition for the user elements in the input file by removing the standard element defined with `*Element` keyword in generating the model.

```
*User Element, Type=< >, Nodes=< >, Coordinates=< >, Properties=<
   >, Iproperties=< >
<list of degrees of freedom>
*Element, type=< >
<list of element nodal connectivity>
```

The second step is to define the properties of the user elements by element sets by removing standard keywords `*Section` and `*Material`.

```
*Uel property, Elset= < >
<list of properties>
```

The third step is optional, but if the user wants to visualize the results in Abaqus/Viewer, a set of overlaying dummy elements needs to be added to the input file.

```
*Element, Type=< >
<list of element nodal connectivity>
*Elset, Elset=elDummy
< >
*Solid section, Elset=elDummy, Material=Dummy
*Material, Name=Dummy
*Elastic
1.e-20
*User output variables
< >
```

The fourth step is to request element-level output in load steps.

```
*Element output, Elset=< >
uvarm
```

3 Source code availability

GitHub repository: https://github.com/bibekanandadatta/Abaqus-UEL-Elasticity.

4 Citation

Datta, B. (2024). *An Abaqus user element (UEL) implementation of linear elastostatics.* Zenodo. https://doi.org/10.5281/zenodo.11075088 (APA format)

Bibliographic references

Dassault Systèmes. (2024). *SIMULIA User Assistance 2024: Abaqus.* https://help.3ds.com

Fish, J., & Belytschko, T. (2007). *A First Course in Finite Elements.* John Wiley & Sons, Inc.

Hughes, T. J. R. (1987). *The Finite Element Method: Linear Static and Dynamic Finite Element Analysis* (1st ed.). Prentice-Hall.

Zienkiewicz, O. C., Taylor, R. L., & Fox, D. (2014). *The Finite Element Method for Solid and Structural Mechanics* (7th ed.). Butterworth-Heinemann.

Zienkiewicz, O. C., Taylor, R. L., & Zhu, J. Z. (2013). *The Finite Element Method: Its Basis and Fundamentals* (7th ed.). Butterworth-Heinemann.

Appendix A. Modifications for plane and axisymmetric problems

For plane (strain or stress) and axisymmetric problems, the nodal coordinate vector, $\mathbf{x}^a$, nodal displacement (degrees of freedom) vector, $\mathbf{u}^a$, and nodal interpolation function matrix, $\mathbf{N}^a_{\mathbf{u}}$, can be represented in the following form.

$$\mathbf{x}^a = [x_1^a,\ x_2^a]^\top, \quad \mathbf{u}^a = [u_1^a,\ u_2^a]^\top, \quad \text{and} \quad \mathbf{N}^a_{\mathbf{u}} = \begin{bmatrix} N_i & 0 \\ 0 & N_i \end{bmatrix}. \tag{A.1}$$

By repeating these sub-vectors ($\mathbf{x}^a$ and $\mathbf{u}^a$) and sub-matrix ($\mathbf{N}^a_{\mathbf{u}}$) for each node, element-level nodal coordinate vector, $\mathbf{x}_e$, displacement (degrees of freedom) vector, $\mathbf{u}_e$, and the interpolation function matrix, $\mathbf{N}_{\mathbf{u}}$, can be obtained as shown previously.

For two-dimensional plane strain and plane stress cases, the matrix form of $[\mathbf{B}^a_{\mathbf{u}}]$ and as well as the vector form of $\boldsymbol{\sigma}$ can be easily reduced as follows,

$$\mathbf{B}^a_{\mathbf{u}} = \begin{bmatrix} N^a_{,1} & 0 \\ 0 & N^a_{,2} \\ N^a_{,2} & N^a_{,1} \end{bmatrix}_{\text{nstress}\times\text{nsd}}, \boldsymbol{\sigma} = \begin{Bmatrix} \sigma_{11} \\ \sigma_{22} \\ \sigma_{12} \end{Bmatrix}. \tag{A.2}$$

On the other hand, for axisymmetric problems, $[\mathbf{B}^a_{\mathbf{u}}]$ and as well as the vector form of $\boldsymbol{\sigma}$ can be written as,

$$\mathbf{B}_{\mathbf{u}}^{a} = \begin{bmatrix} N_{,1}^{a} & 0 \\ 0 & N_{,2}^{a} \\ \frac{N^{a}}{r} & 0 \\ N_{,2}^{a} & N_{,1}^{a} \end{bmatrix}_{\text{nstress}\times\text{nsd}}, \boldsymbol{\sigma} = \begin{Bmatrix} \sigma_{rr} \\ \sigma_{zz} \\ \sigma_{\theta\theta} \\ \sigma_{rz} \end{Bmatrix}. \tag{A.3}$$

The element-level strain-displacement matrix, $\mathbf{B}_{\mathbf{u}}$, can constructed by repeating these nodal-level sub-matrices.

In finite element implementation, it is perhaps convenient to calculate the stress tensor (3×3) and elasticity tensor $(3 \times 3 \times 3 \times 3)$ for a general three-dimensional case first. Consequently, by applying the Voigt notation convention, the vector form of the Cauchy stress tensor (6×1) and the Voigt matrix form of the elasticity tensor (6×6) can be obtained. If necessary, further reduction can be performed to obtain the reduced form of Voigt matrix and Cauchy stress vector for axisymmetric problems ((4×4) and (4×1), respectively) for plane strain problems ((3×3) and (3×1), respectively).

For three-dimensional isotropic linear elastic materials, using Voigt notation, Hooke's law can be written as,

$$\begin{Bmatrix} \sigma_{11} \\ \sigma_{22} \\ \sigma_{33} \\ \sigma_{23} \\ \sigma_{13} \\ \sigma_{12} \end{Bmatrix} = \begin{bmatrix} 2\mu+\lambda & \lambda & \lambda & 0 & 0 & 0 \\ \lambda & 2\mu+\lambda & \lambda & 0 & 0 & 0 \\ \lambda & \lambda & 2\mu+\lambda & 0 & 0 & 0 \\ 0 & 0 & 0 & \mu & 0 & 0 \\ 0 & 0 & 0 & 0 & \mu & 0 \\ 0 & 0 & 0 & 0 & 0 & \mu \end{bmatrix} \begin{Bmatrix} \varepsilon_{11} \\ \varepsilon_{22} \\ \varepsilon_{33} \\ 2\varepsilon_{23} \\ 2\varepsilon_{13} \\ 2\varepsilon_{12} \end{Bmatrix} \tag{A.4}$$

For axisymmetric problems ((r, z, θ)-coordinate), out-of-plane shear strains are zero ($\varepsilon_{r\theta} = \varepsilon_{z\theta} = 0$) but the out-of-plane axial strain is non-zero ($\varepsilon_{\theta\theta} \neq 0$). Thus, the Hooke's law appears as follows,

$$\begin{Bmatrix} \sigma_{rr} \\ \sigma_{zz} \\ \sigma_{\theta\theta} \\ \sigma_{rz} \end{Bmatrix} = \frac{E}{(1-2\nu)(1+\nu)} \begin{bmatrix} 1-\nu & \nu & \nu & 0 \\ \nu & 1-\nu & \nu & 0 \\ \nu & \nu & 1-\nu & 0 \\ 0 & 0 & 0 & \frac{1-2\nu}{2} \end{bmatrix} \begin{Bmatrix} \varepsilon_{rr} \\ \varepsilon_{zz} \\ \varepsilon_{\theta\theta} \\ 2\varepsilon_{rz} \end{Bmatrix}. \tag{A.5}$$

For plane strain cases in which the out-of-plane strain components are zero, *i.e.*, $\varepsilon_{33} = \varepsilon_{13} = \varepsilon_{31} = \varepsilon_{23} = \varepsilon_{32} = 0$. Thus, I have,

$$\begin{Bmatrix} \sigma_{11} \\ \sigma_{22} \\ \sigma_{12} \end{Bmatrix} = \frac{E}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1-\nu & \nu & 0 \\ \nu & 1-\nu & 0 \\ 0 & 0 & \frac{1-2\nu}{2} \end{bmatrix} \begin{Bmatrix} \varepsilon_{11} \\ \varepsilon_{22} \\ 2\varepsilon_{12} \end{Bmatrix}. \tag{A.6}$$

In the case of plane strain analyses, out-of-plane axial stress is not zero ($\sigma_{33} \neq 0$), rather it depends on the in-plane axial stress components.

Constitutive calculation for the plane stress cases should be considered separately since $\varepsilon_{33} \neq 0$, rather it requires enforcing the condition $\sigma_{33} = 0$ to obtain the reduced form of Voigt matrix, $\mathbf{D}$. For a material with anisotropic linear elastic behavior under the plane stress condition, the following steps can be used Hughes, 1987; Zienkiewicz et al., 2013.

1. First, obtain the reduced Voigt matrix, $\mathbf{D}$, for a general two-dimensional case (equivalent to the axisymmetric case). The first three rows and columns correspond to three axial stress and strain components and the fourth row and column correspond to the in-plane shear stress and strain component. By enforcing the plane stress condition ($\sigma_{33} = 0$), modify the two-dimensional Voigt matrix to obtain the following form, $\hat{\mathbf{D}}$,

$$\mathbf{D} = \begin{bmatrix} D_{11} & D_{12} & D_{13} & D_{14} \\ D_{21} & D_{22} & D_{23} & D_{24} \\ D_{31} & D_{32} & D_{33} & D_{34} \\ D_{41} & D_{42} & D_{43} & D_{44} \end{bmatrix} \xrightarrow[\text{modification}]{\sigma_{33}=0} \hat{\mathbf{D}} = \begin{bmatrix} \hat{D}_{11} & \hat{D}_{12} & 0 & \hat{D}_{14} \\ \hat{D}_{21} & \hat{D}_{22} & 0 & \hat{D}_{24} \\ 0 & 0 & 0 & 0 \\ \hat{D}_{41} & \hat{D}_{42} & 0 & \hat{D}_{44} \end{bmatrix}, \tag{A.7}$$

$$\text{where,} \quad \hat{D}_{ij} = D_{ij} - D_{i3} D_{33}^{-1} D_{3j}.$$

2. Remove the zero rows and columns (third row and third column) to obtain the final form reduced Voigt matrix, $\hat{\mathbf{D}}$, for plane stress analysis. In the case of plane stress analyses, out-of-plane axial strain is not zero ($\varepsilon_{33} \neq 0$), rather it depends on the in-plane axial strain components.

$$\hat{\mathbf{D}} = \begin{bmatrix} \hat{D}_{11} & \hat{D}_{12} & \hat{D}_{14} \\ \hat{D}_{21} & \hat{D}_{22} & \hat{D}_{24} \\ \hat{D}_{41} & \hat{D}_{42} & \hat{D}_{44} \end{bmatrix} \tag{A.8}$$

For example, for an isotropic linear elastic materials under the plane stress condition, by performing the above steps, following form of Hooke's law can be obtained,

$$\begin{Bmatrix} \sigma_{11} \\ \sigma_{22} \\ \sigma_{12} \end{Bmatrix} = \frac{E}{1-\nu^2} \begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \frac{1-\nu}{2} \end{bmatrix} \begin{Bmatrix} \varepsilon_{11} \\ \varepsilon_{22} \\ 2\varepsilon_{12} \end{Bmatrix}. \tag{A.9}$$

For axisymmetric problems, the element stiffness matrix, $\mathbf{k}^e_{\mathbf{uu}}$, and the element residual vector, $\mathbf{R}^e_{\mathbf{u}}$, needs to be modified as follows,

$$\begin{aligned} \mathbf{k}^e_{\mathbf{uu}} &= 2\pi \int_{\Omega^e} \mathbf{B}_{\mathbf{u}}^\top \mathbf{D} \mathbf{B}_{\mathbf{u}} \; r \; drdz, \\ \mathbf{R}^e_{\mathbf{u}} &= -2\pi \int_{\Omega^e} \mathbf{B}_{\mathbf{u}}^\top \boldsymbol{\sigma} \; rdrdz + 2\pi \int_{\Omega^e} \rho \mathbf{N}_{\mathbf{u}}^\top \mathbf{b}^e \; r \; drdz + 2\pi \int_{\Gamma^e_t} \mathbf{N}_{\mathbf{u}}^\top \mathbf{t}^e \; r \; dz. \end{aligned} \tag{A.10}$$