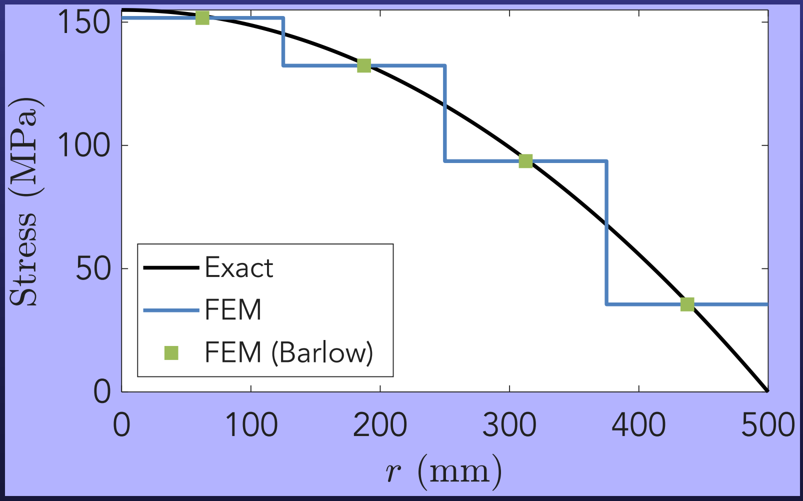
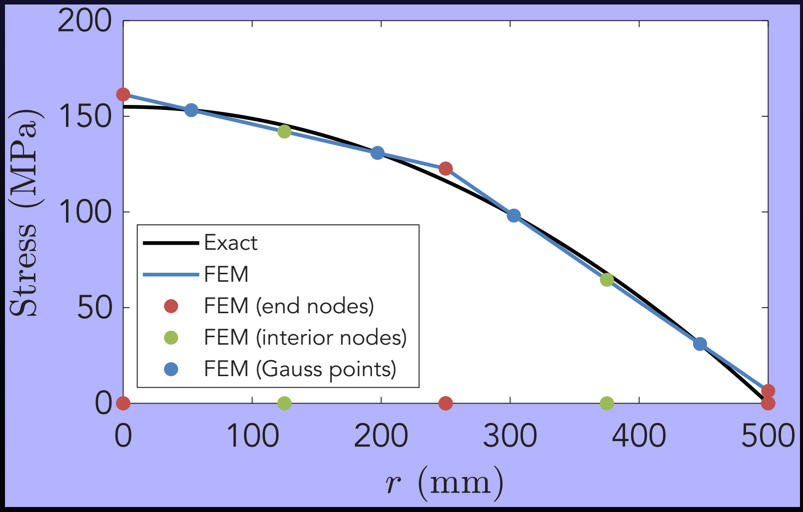
Finite Element Analysis of a Rotating Elastic Rod

Comparison of Linear and Quadratic Elements

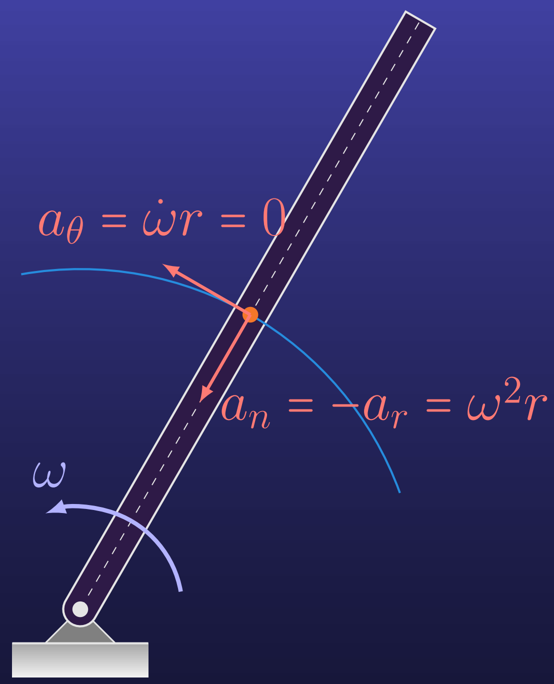


Linear Elements Piecewise Constant Stress approximation.



Quadratic Elements Piecewise Discontinuous Linear Stress approximation.

Centrifugal Load on Rotating Elastic Bar



Linear varying Centrifugal force per unit length $f_r(r) = \rho A a_r = \rho A(\omega^2 r)$ for constant angular speed ω in circular motion about the fixed pivot.

- The uniform elastic steel rod has length $L = 50$ cm, section area $A = 4$ mm^2, rotating about a fixed pivot with constant angular speed ω in rad/sec corresponding to 120 rpm.
- Let's study an interesting Finite Element Analysis (FEA) of a rotating elastic rod using both linear and quadratic elements and verify the approximate solution with an exact solution.

Problem Statement

1. Consider a uniform rod with length L rotating in the horizontal plane about a fixed pivot. The load is the centrifugal mass inertia force due to inward radial acceleration for circular motion with constant rotational speed.

2. Compare finite element solutions with mesh refinement for displacement and axial stress comparing connected meshes of 2-node, linear elements, and 3-node, quadratic elements, ensuring the same number of total nodes.

3. Verify the FE approximate solutions with the exact analytical solution.

Exact Solution

- Since the centrifugal force at an element of material in the rod $f_r(r) = \rho A a_r = \rho A \omega^2 r$, increases linearly with radius r, we cannot assume the internal force and stress is constant everywhere in the rod.

- Instead, since the inertial force is linear, the internal axial force and stress will vary quadratically with r, and for a linear elastic material, the strain also varies quadratically.

- To find displacement, integrate the quadratic strain resulting in a cubic displacement.

- From Newton's law of motion, the strong differential boundary value problem for the rotating elastic rod can be summarized as: Given the centrifugal body load $f_r(r) = \rho A \omega^2 r$; Find $u(r)$ such that

$$\frac{d}{dr}\left(EA\frac{du}{dr}\right) = -f_r$$

subject to the boundary conditions,

$$u(0) = 0, \quad N(L) = EA\left.\frac{du}{dr}\right|_{r=L} = 0$$

- This equation can be solved by integrating twice and enforcing the boundary conditions.
- The axial (normal) stress is a quadratic polynomial in r:

$$\boxed{\sigma(r) = \frac{N(r)}{A} = \frac{\rho\omega^2}{2}\left(L^2 - r^2\right)}$$

- This is the tensile stress due to centrifugal mass inertia, increasing from 0 at the free end $r = L$ and to a maximum at the fixed end $r = 0$.

$$\sigma_{\max} = \sigma(0) = \frac{N(r)}{A} = \frac{\rho(\omega L)^2}{2}$$

The corresponding axial displacement is the cubic polynomial:

$$\boxed{u(r) = \frac{\rho\omega^2 r}{6E}\left(3L^2 - r^2\right)}$$

- Since the centrifugal force was linear, the internal force, stress, and strain were quadratic. Thus, the displacement varies as a cubic polynomial in the nondimensional radius r/L.

Finite Element Approximation

Weak form of the Boundary Value Problem and the Principle of Virtual Displacements

- The finite element approximation satisfies the corresponding weak form of the boundary value problem equivalent to the principle of virtual work for stress analysis.

- Find $u(r)$ such that $u(0) = 0$ for all virtual displacements δu with the constraint $\delta u(0) = 0$ (to remove unknown reaction forces from the problem statement), such that

$$\int_0^L \frac{d\delta u}{dr} EA \frac{du}{dr}\, dr = \int_0^L \delta u\, f_r\, dr$$

- After finite element discretization and C^0 approximation with Lagrange interpolation (shape) function for u and δu within elements, and after enforcing displacement compatibility for assembly, we solve the algebraic system of discrete nodal equilibrium equations:

$$[\boldsymbol{K}]\{\boldsymbol{u}\} = \{\boldsymbol{F}\}$$

- The left side of the integral equation gives $[\boldsymbol{K}]\{\boldsymbol{u}\}$, where $\boldsymbol{K}$ is the assembled global symmetric stiffness matrix and $\boldsymbol{u}$ is a vector array of nodal displacements (degrees-of-freedom).
- The right side of the integral variational equation results in the assembled load vector $\{\boldsymbol{F}\}$ due to work equivalent element nodal load vector contributions from the centrifugal load f_r.

Element stiffness matrix

- The element stiffness matrix has the form,

$$\boldsymbol{k} = \int_{r_1}^{r_2} \boldsymbol{B}^T E A \boldsymbol{B} \, dr$$

where $\boldsymbol{B}$ is the array relating axial strain to nodal displacements – the elements of this matrix are derivatives of the element shape functions.

  - For 2-node elements, these shape functions are linear polynomials.
  - For 3-node elements, these shape functions are quadratic polynomials.

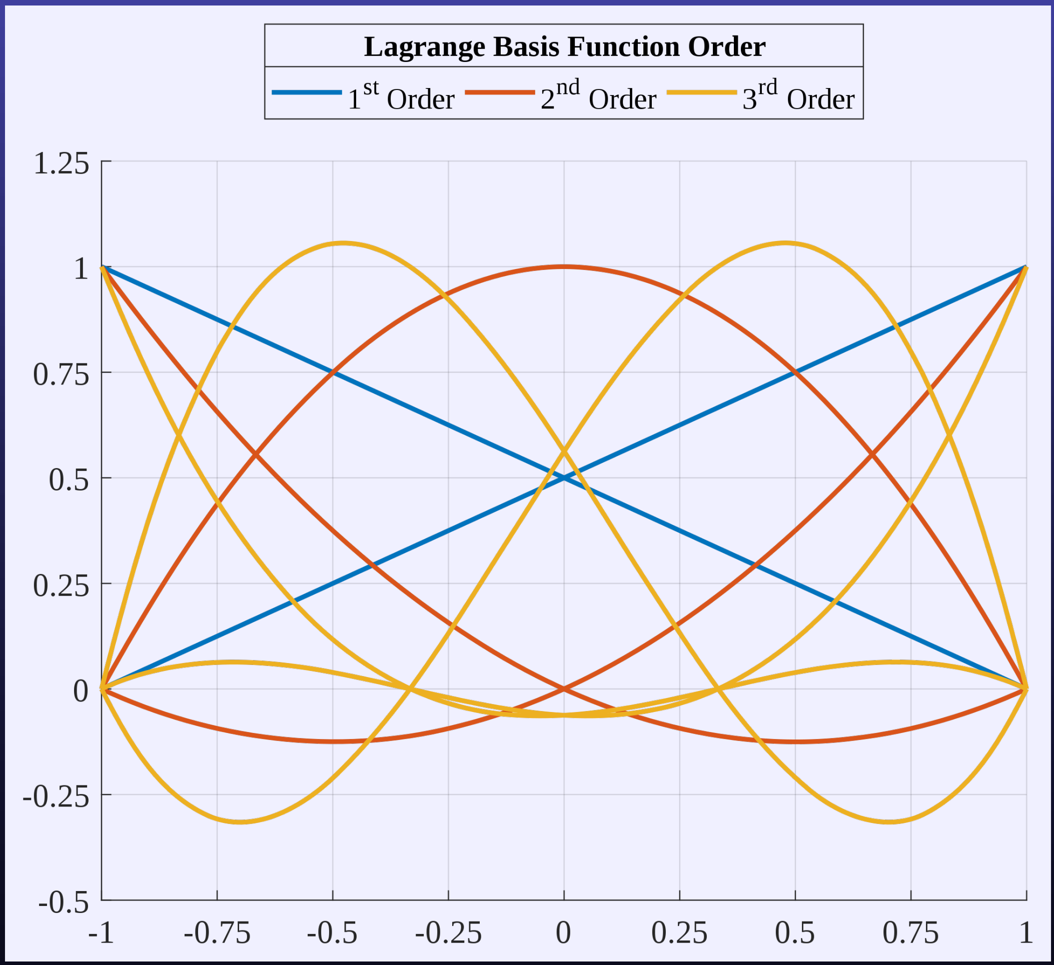
Element nodal load vector

- The element nodal load vector takes the form,

$$\boldsymbol{f} = \int_{r_1}^{r_2} \boldsymbol{N}^T f_r(r)\, dr$$

where $f_r(r) = \rho A \omega^2 r$ is the centrifugal load per length, and $\boldsymbol{N}$ is an array of nodal shape functions.

High-Order Finite Element Shape Functions



Lagrange interpolation functions of order one (linear), two (quadratic), and three (cubic) used for approximation of displacement within an element.

- The element nodal shape functions N have the interpolation property of unit value at the defining node and zero at other nodes, ensuring continuity between connected elements.

- Derivatives are *discontinuous* with a jump in values at connecting nodes in a mesh of finite elements. As a result, finite element approximations for strain $\varepsilon = \frac{du}{dx}$ and stress $\sigma = E\frac{du}{dx}$ are discontinuous between connected elements.
  - For 2-node linear elements, the strain and stress are constant within elements.
  - For 3-node quadratic elements, the strain and stress vary linearly within elements.

Element Stiffness Matrices for 2-node linear and 3-node quadratic elements

- The element stiffness matrices are obtained from integration over the element.
  - The uniform rod element stiffness matrix for the 2-node linear element is:

$$[\boldsymbol{k}] = k_e \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$$

  - The uniform rod element stiffness matrix for the 3-node quadratic element is:

$$[\boldsymbol{k}] = \frac{k_e}{3} \begin{bmatrix} 7 & -8 & 1 \\ -8 & 16 & -8 \\ 1 & -8 & 7 \end{bmatrix}$$

where

$$k_e = \frac{EA}{\ell_e}$$

is the axial 'spring' stiffness for an elastic bar member of length ℓ_e defined in terms of the two end nodes of the element.

- In the case of the 2-node linear element $\ell_e = r_2 - r_1$.
- In the case of the 3-node quadratic element with midpoint node 2 and end nodes 1 and 3: $\ell_e = r_3 - r_1$.

Nodal force vector equivalent to linear load

Load vector for 2-node linear element

- For a linear body force distribution, $f_r(r)$ like our case of Centrifugal load per unit length, after integration over the element, the work equivalent nodal loads for the 2-node linear element are:

$$\{\boldsymbol{f}\} = \frac{\ell_e}{6}\begin{Bmatrix} 2f_r(r_1) + f_r(r_2) \\ f_r(r_1) + 2f_r(r_2) \end{Bmatrix}$$

where $f_r(r_1)$ and $f_r(r_2)$ are the body force function evaluations at the first and last node of the element connected to other elements in the mesh.

In our case of Centrifugal load,

$$f(r_1) = \rho A \omega^2 r_1, \quad f(r_2) = \rho A \omega^2 r_2$$

giving the nodal loads,

$$\{\boldsymbol{f}\} = \frac{m_e \omega^2}{6}\begin{Bmatrix} 2r_1 + r_2 \\ r_1 + 2r_2 \end{Bmatrix}$$

where $m_e = \rho A \ell_e$ is the mass for the uniform rod element.

Load vector for 3-node quadratic element

- For a linear body force distribution, $f_r(r)$ like our case of Centrifugal load per unit length, after integration over the element, the work equivalent nodal loads for the 3-node quadratic element are:

$$\{\boldsymbol{f}\} = \frac{\ell_e}{6}\left\{\begin{matrix} f_r(r_1) \\ 2f_r(r_1) + 2f(r_3) \\ f_r(r_3) \end{matrix}\right\}$$

where $f_r(r_1)$ and $f_r(r_3)$ are the body force function evaluations at the first and last (3rd) nodes of the 3-node element connected to other elements in the mesh. The midpoint node 2 is internal to the quadratic element and not directly connected to other elements in the mesh (The shape function associated with this midpoint node is zero at the end connecting nodes).

- In our case of Centrifugal load,

$$f(r_1) = \rho A\omega^2 r_1, \quad f(r_3) = \rho A\omega^2 r_3$$

giving the nodal loads,

$$\{\boldsymbol{f}\} = \frac{m_e\omega^2}{6}\left\{\begin{matrix} r_1 \\ 2(r_1 + r_3) \\ r_3 \end{matrix}\right\}$$

Finite Element Solution using a uniform mesh of two, 2-node linear elements

Let's find the finite element solution using just two elements by hand and compare it to the exact solution for displacement and stress.

- The assembled load vector for the mesh of two connected elements:

$$\begin{Bmatrix} f_1^{[1]} \\ f_2^{[1]} + f_1^{[2]} \\ f_2^{[2]} \end{Bmatrix} = \frac{m_e \omega^2}{6} \begin{Bmatrix} 2r_1 + r_2 \\ r_1 + 4r_2 + r_3 \\ r_2 + 2r_3 \end{Bmatrix}$$

$$\left[\begin{array}{c|cc} k_e & -k_e & 0 \\ \hline -k_e & 2k_e & -k_e \\ 0 & -k_e & k_e \end{array}\right] \begin{Bmatrix} \cancelto{0}{u_1} \\ u_2 \\ u_3 \end{Bmatrix} = \frac{m_e \omega^2}{6} \begin{Bmatrix} 2r_1 + r_2 \\ r_1 + 4r_2 + r_3 \\ r_2 + 2r_3 \end{Bmatrix} + \begin{Bmatrix} \bar{F}_1 \\ 0 \\ 0 \end{Bmatrix}$$

where $\bar{F}_1$ is a reaction force at the fixed pivot point at mesh node 1.

- Enforcing the essential boundary condition $u(0) = u_1 = 0$, to remove rigid body motions and using the virtual displacement constraint $\delta u(0) = 0$, to eliminate the first-row equation with the unknown external reaction force $\bar{F}_1$, we have the reduced, symmetric positive-definite and invertible *reduced partitioned system*:

$$k_e \begin{bmatrix} 2 & -1 \\ -1 & 1 \end{bmatrix} \begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} = \frac{m_e \omega^2}{6} \begin{Bmatrix} r_1 + 4r_2 + r_3 \\ r_2 + 2r_3 \end{Bmatrix}$$

- For the two element mesh, the coordinates are $r_1 = 0$, $r_2 = \ell = L/2$, and $r_3 = 2\ell = L$.

$$k_e \begin{bmatrix} 2 & -1 \\ -1 & 1 \end{bmatrix} \begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} = \frac{m_e \omega^2 \ell}{6} \begin{Bmatrix} 6 \\ 5 \end{Bmatrix}$$

- Solving by taking the inverse,

$$\begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} = \begin{bmatrix} 1 & 1 \\ 1 & 2 \end{bmatrix} \begin{Bmatrix} 6 \\ 5 \end{Bmatrix} \frac{m_e \omega^2 \ell}{6k_e} = \begin{Bmatrix} 11 \\ 16 \end{Bmatrix} \frac{m_e \omega^2 \ell}{6k_e}$$

With $m_e = \rho A \ell$, and $k_e = EA/\ell$,

$$\begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} 11 \\ 16 \end{Bmatrix} \frac{\rho \omega^2 \ell^3}{6E}$$

with $\ell = L/2$,

$$\begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} 11 \\ 16 \end{Bmatrix} \frac{\rho\omega^2 L^3}{48E}$$

- Simplifying, and including the boundary condition $u_1 = 0$, the FEM nodal displacements are:

$$\begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 11/48 \\ 1/3 \end{Bmatrix} \frac{\rho\omega^2 L^3}{E}$$

- Let's check these FEM nodal displacements with the exact cubic polynomial solution $u(r)$ evaluated at the node coordinates.

$$u(r_1) = u(0) = 0$$

$$u(r_2) = u(L/2) = \frac{\rho\omega^2 (L/2)}{6E}\left(3L^2 - (L/2)^2\right) = \frac{11\rho\omega^2 L^3}{48E}$$

$$u(r_3) = u(L) = \frac{\rho\omega^2 L}{6E}\left(3L^2 - L^2\right) = \frac{\rho\omega^2 L^3}{3E}$$

- We just found that (even with just two elements) the finite element nodal displacements are EXACT for this problem!

$$\begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} u(r_1) \\ u(r_2) \\ u(r_3) \end{Bmatrix} = \begin{Bmatrix} 0 \\ 11/48 \\ 1/3 \end{Bmatrix} \frac{\rho\omega^2 L^3}{E}$$

- While this might appear surprising since, in general, the FEM is an approximate numerical solution to the weak form of the boundary value problem for our linear elastic model or a rotating elastic rod with constant angular speed, this result was to be expected from a mathematical analysis of this type of problem. In a rigorous mathematical proof that the finite element solution is exact at the nodes for any body load like our linear varying Centrifugal force under the condition that the rod is uniform with constant EA, where E is Young's modulus and A is the section area.

- Strang and Fix (yes, that famous Gilbert Strang from MIT linear algebra YouTube videos and textbooks) attribute this proof to Douglas and Dupoint. As stated by T.J.R. Hughes in his book The Finite Element Method: Linear Static and Dynamic Finite Element Analysis,

  "results of this kind, embodying exceptional accuracy characteristics, are often referred to as **superconvergence** phenomena. However, the reader should appreciate that, in more complicated situations, we will not be able, in practice, to guarantee nodal exactness. Nevertheless, as we shall see later on, weighted residual procedures provide a framework within which optimal accuracy properties of some sort may often be guaranteed."

- I don't know about you, but this is reassuring to know FEM solutions often guarantee optimal accuracy in some sense. How is it that most finite element books never show this? Do the authors know this?

- **Exercise for the reader:** Show that the finite element nodal displacement is exact using just a single element.

Stress Recovery

- While displacement is exact at the nodes and 2nd-order accurate at points within the element, the derivatives, and thus strain and stress, are in general only 1st-order accurate.
- In general, derivatives of a function are an order less accurate than the function.
- However, using the Taylor series and the mean value theorem of Calculus, it can be shown that at the mid-points of the 2-node linear element, the derivates are 2nd-order and higher-order accurate than the 1st-order at other points.
- If the exact displacement solution $u(r)$ is linear or quadratic, then the derivative and stress are exact at midpoints; this is the case when a distributed body force is constant.
- However, for our Centrifugal force $f_r = \rho\omega^2 r$, the load is linear, not constant, and the displacement is cubic.
- As a result, the stress will not be exact at the midpoint, but the stress at the midpoint is second-order accurate compared to only first-order at other points.

- As pointed out by Hughes, 'The midpoints of linear elements are called the **Barlow stress points**, after Barlow, who first noted that points of optimal accuracy existed within elements.'
- Let's use our nodal displacement solutions to recover stresses.
- The displacement within an element is the linear interpolation of nodal values.
- The strain at a point is the derivative (gradient) of displacement at that point.
- Since the displacement is linear, the derivative gives constant strain, and if E is constant from Hooke's law, the stress is also constant.

$$\varepsilon_e = \frac{du_e}{dx} = \boldsymbol{B}_e\,\boldsymbol{u}_e = \frac{u_2^e - u_1^e}{\ell},$$
$$\sigma_e = E\varepsilon_e = \boldsymbol{B}_e\,\boldsymbol{u}_e = E\left(\frac{u_2^e - u_1^e}{\ell}\right)$$

- The strains and stresses are piecewise constant and discontinuous at connected mesh nodes.

- For our two elements,

$$\sigma_1 = E\left(\frac{u_2 - u_1}{\ell}\right) = \frac{11}{24}\rho(\omega L)^2$$
$$\sigma_2 = E\left(\frac{u_3 - u_2}{\ell}\right) = \frac{5}{24}\rho(\omega L)^2$$

- Let's check these FEM stress approximations with the exact stress solution given by the quadratic polynomial solution $\sigma(r)$ evaluated at the node coordinates.
  - For Element $[1]$, at the midpoint $r = \ell/2 = L/4$,

$$\sigma(L/4) = \frac{\rho\omega^2}{2}\left(L^2 - (L/4)^2\right) = \frac{15}{32}\rho(\omega L)^2$$

  - For Element $[2]$, with midpoint $r = 3\ell/2 = 3L/4$,

$$\sigma(3L/4) = \frac{\rho\omega^2}{2}\left(L^2 - (3L/4)^2\right) = \frac{7}{32}\rho(\omega L)^2$$

These don't match the finite element stress values.

- Let's check the error at the midpoints of the two elements.
- At the midpoints of elements [1] and [2], the percent error is:

$$\left(\frac{\sigma_1}{\sigma(L/4)} - 1\right) \times 100\% = -\frac{1}{45} \times 100\% = 2.2\%$$
$$\left(\frac{\sigma_2}{\sigma(3L/4)} - 1\right) \times 100\% = -\frac{1}{21} \times 100\% = 4.8\%$$

Wow, Barlow was right; this is incredible stress accuracy with a mesh of only two 2-node linear elements!

Nodal Stress Smoothing by Interpolation

- Since the finite element stress solution is discontinuous with a jump, there is no unique solution at the connecting node.
- Let's do some stress smoothing using simple interpolation; this results in a simple average of the two-element constant solutions averaged at the connecting node.

$$\sigma_{avg} = \frac{\sigma_1 + \sigma_2}{2} = \frac{\rho(\omega L)^2}{3}$$

- Let's compare it to the exact stress evaluated at node 2.

$$\sigma(r_2) = \sigma(L/2) = \frac{3\rho(\omega L)^2}{8}$$

This gives an error,

$$\left(\frac{\sigma_{avg}}{\sigma(L/2)} - 1\right) \times 100\% = -\frac{1}{9} \times 100\% = 11\%$$

It's not too bad, but we need mesh refinement with more elements for improved accuracy.

Finite Element Solution using a single 3-node quadratic element

Now, let's examine the accuracy using just a single 3-node quadratic element.

$$\frac{k_e}{3}\left[\begin{array}{r|rr} 7 & -8 & 1 \\ \hline -8 & 16 & -8 \\ 1 & -8 & 7 \end{array}\right]\begin{Bmatrix} \cancelto{0}{u_1} \\ u_2 \\ u_3 \end{Bmatrix} = \frac{m_e\omega^2}{6}\begin{Bmatrix} r_1 \\ 2(r_1+r_3) \\ r_3 \end{Bmatrix} + \begin{Bmatrix} \bar{F}_1 \\ 0 \\ 0 \end{Bmatrix}$$

$$\frac{k_e}{3}\begin{bmatrix} 16 & -8 \\ -8 & 7 \end{bmatrix}\begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} = \frac{m_e\omega^2}{6}\begin{Bmatrix} 2(r_1+r_3) \\ r_3 \end{Bmatrix}$$

For the 3-node quadratic element with middle node 2, the element length is $\ell = r_3 - r_1 = L$.

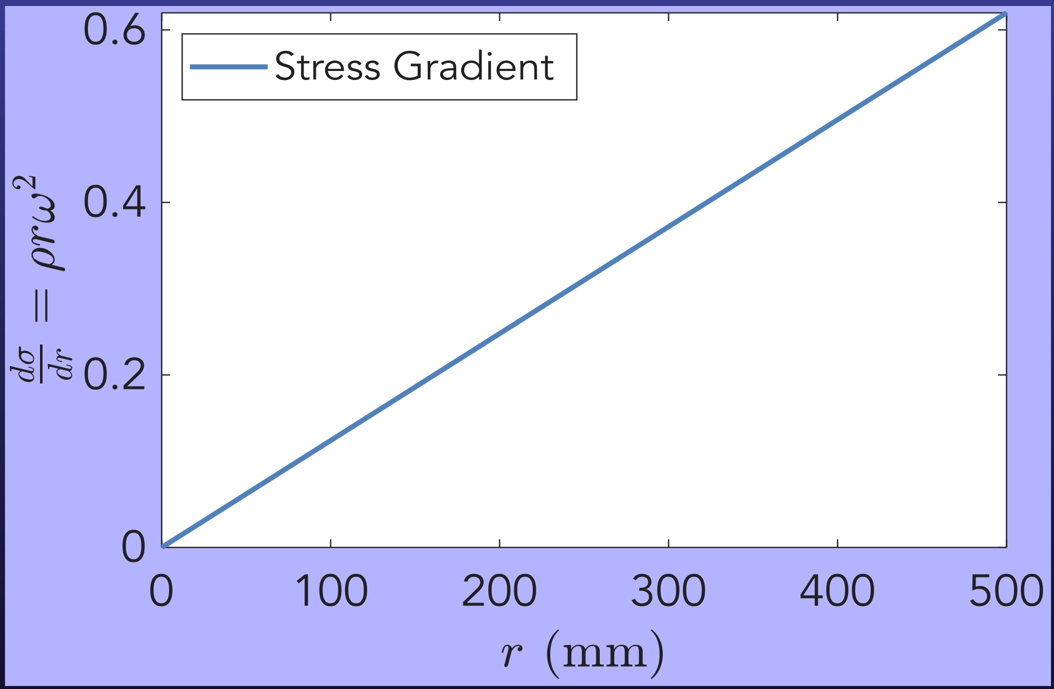
$$\frac{k_e}{3}\begin{bmatrix} 16 & -8 \\ -8 & 7 \end{bmatrix}\begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} = \frac{m_e\omega^2\ell}{6}\begin{Bmatrix} 2 \\ 1 \end{Bmatrix}$$

Solving we find the nodal displacements are EXACT.

$$\begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} = \frac{1}{48}\begin{bmatrix} 7 & 8 \\ 8 & 16 \end{bmatrix}\begin{Bmatrix} 2 \\ 1 \end{Bmatrix}\frac{m_e\omega^2\ell}{2k_e} = \begin{Bmatrix} 11 \\ 16 \end{Bmatrix}\frac{\rho\omega^2L^3}{48E}$$

- We leave it as an exercise for the reader to show that the finite element stress solution for this 3-node quadratic element is higher-order accurate at the 2-point Gauss-Legendre quadrature points within the element.

Deciding on an optimal finite element mesh of connected elements



Analytical Stress Gradient

The exact stress gradient is proportional to the linear centrifugal load

$$\frac{d\sigma}{dr} = \rho\omega^2 r$$

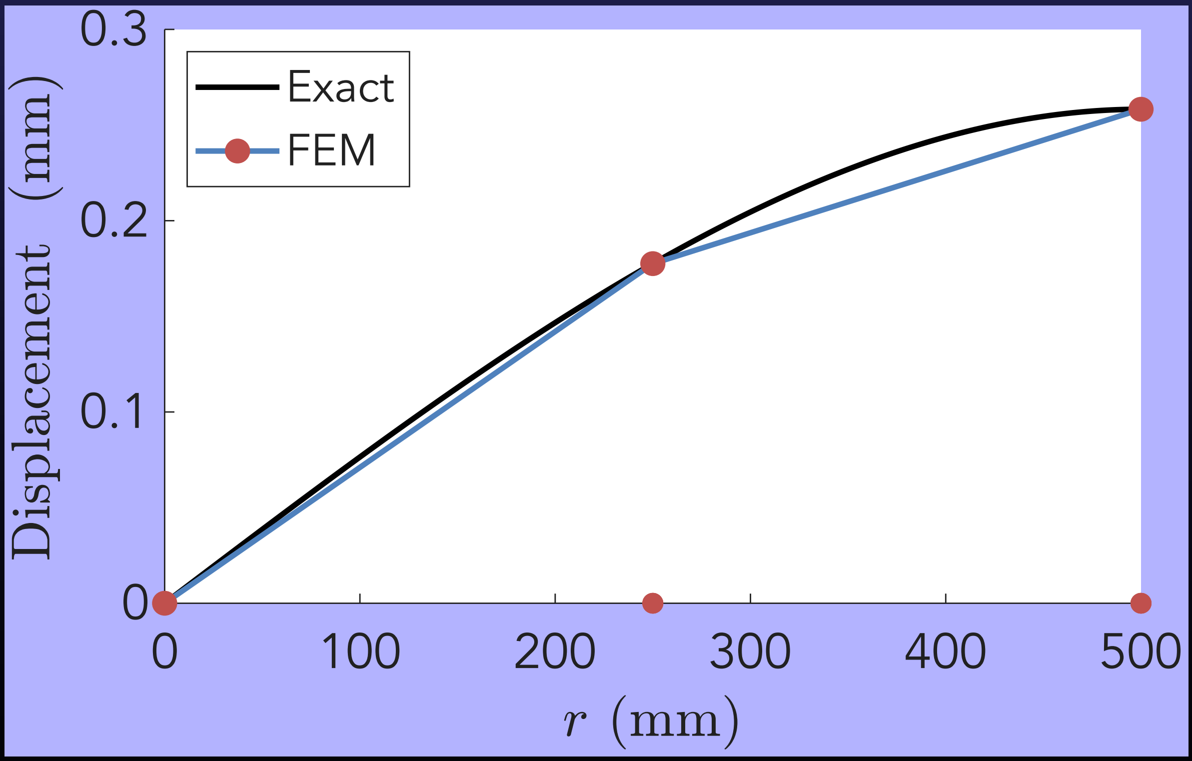
increases linearly, with the second gradient a constant. This implies a linear gradient finite element mesh is optimal. However, for our studies, let's keep things simple and only consider uniform meshes of equally spaced nodes.

Uniform Mesh of Linear Elements

Let's do a mesh refinement study of 2, 4, and 8 linear elements with 3, 5, and 9 mesh nodes, respectively.

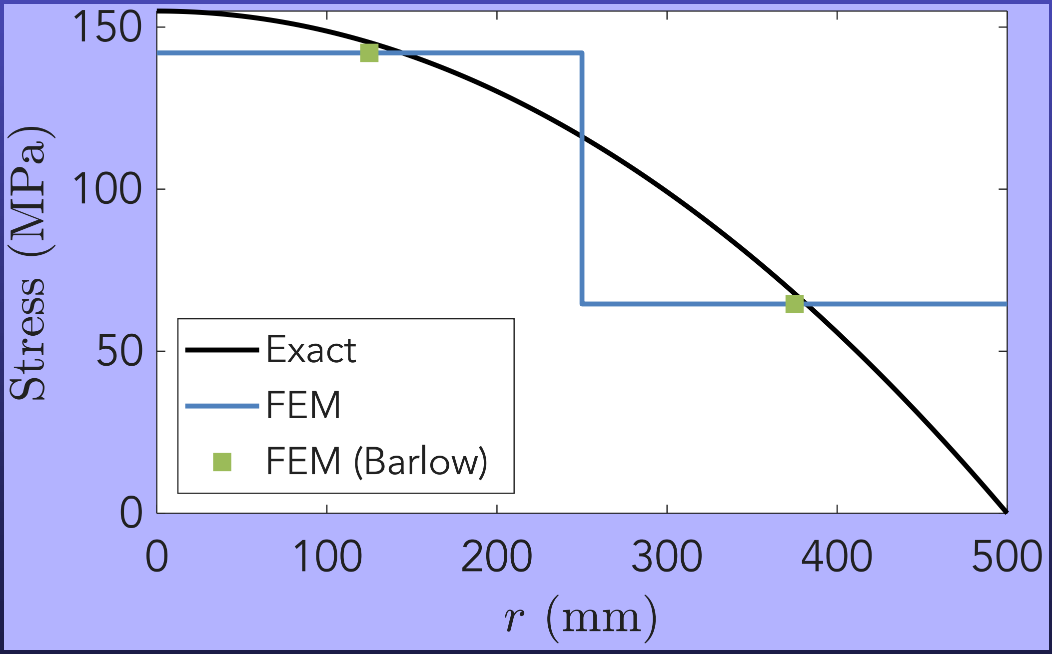
Two Linear Elements

Consider a uniform mesh of a very coarse mesh of only two 2-node linear elements.

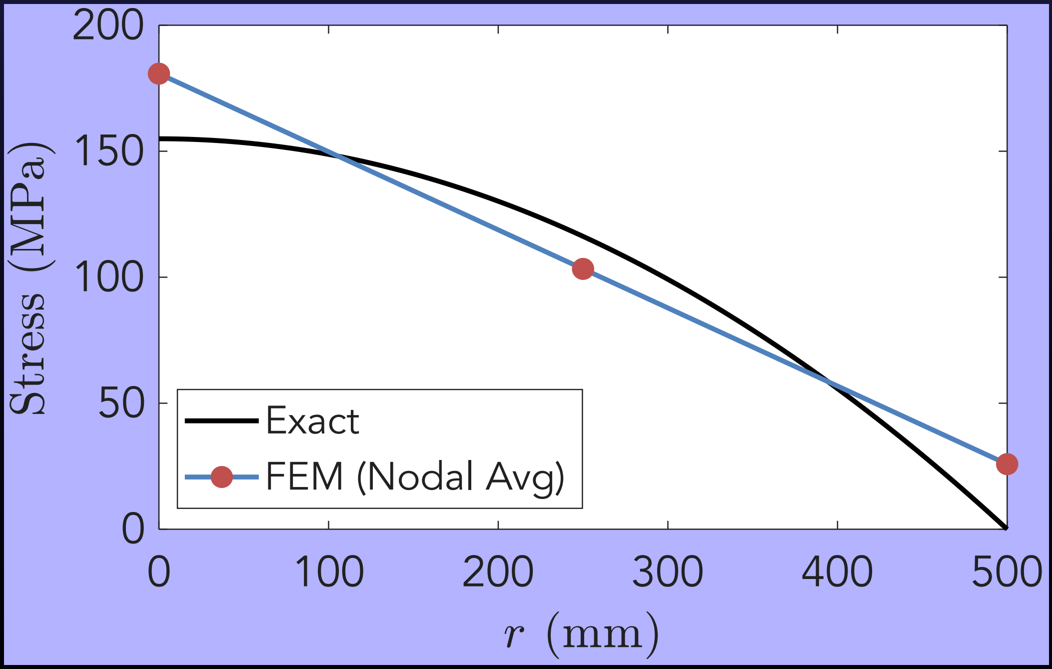


Piecewise Linear Displacement FEM approximation

Stress



Piecewise Constant Stress FEM approximation.

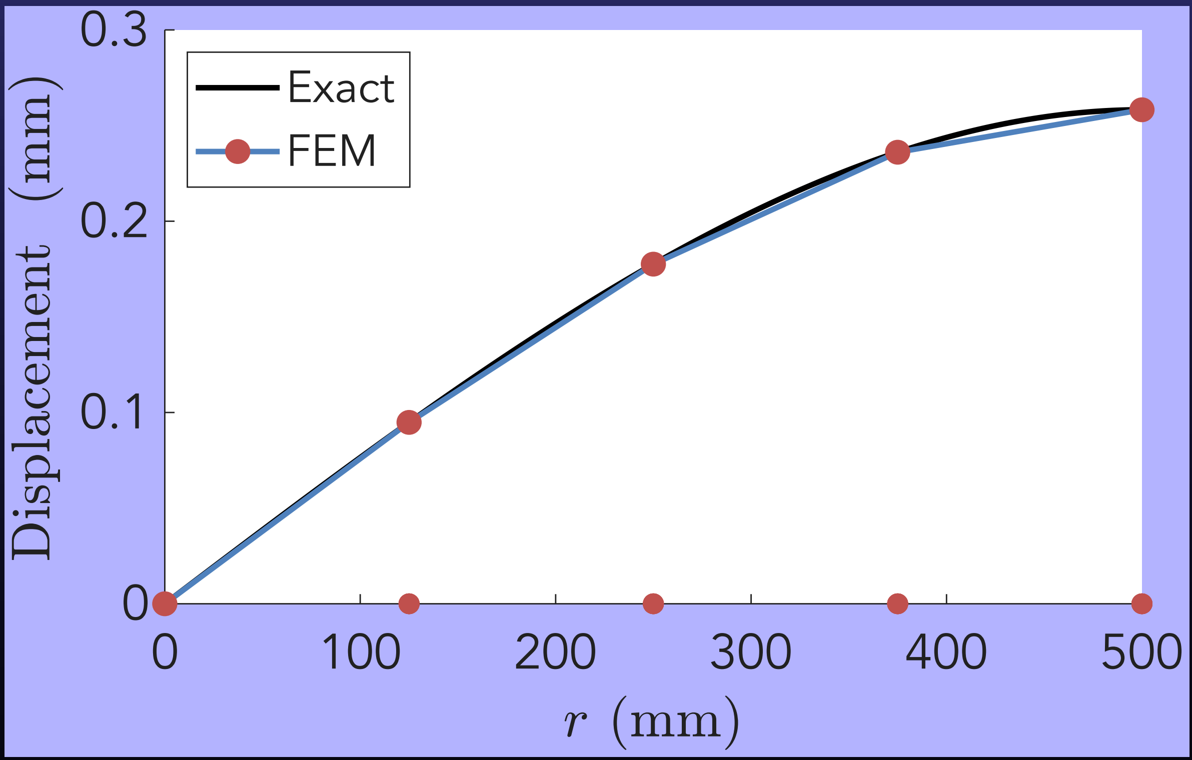


FEM Nodal Stress Smoothing

Four Linear Elements

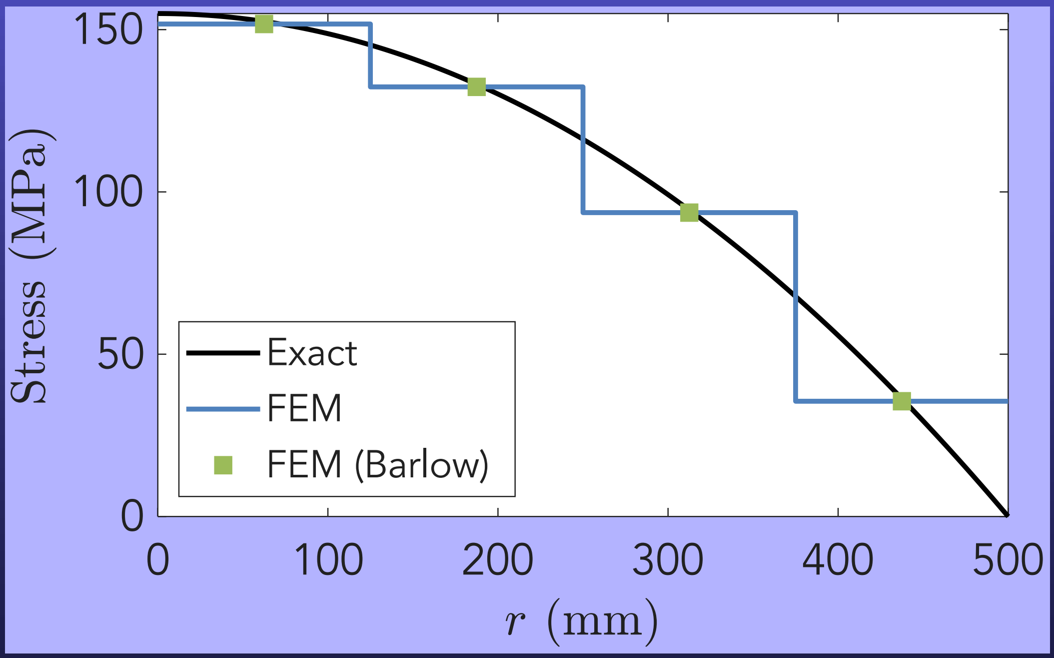
Consider a uniform mesh of a relatively coarse mesh of four 2-node linear elements.

Displacement

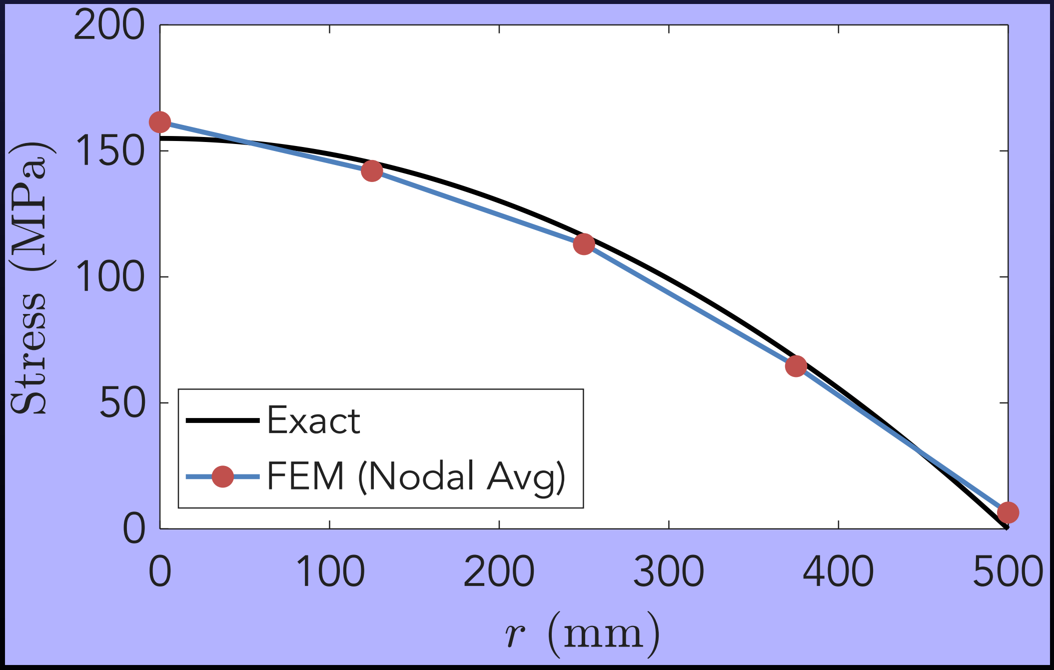


Piecewise Linear Displacement FEM approximation

Stress



Piecewise Constant Stress FEM approximation.

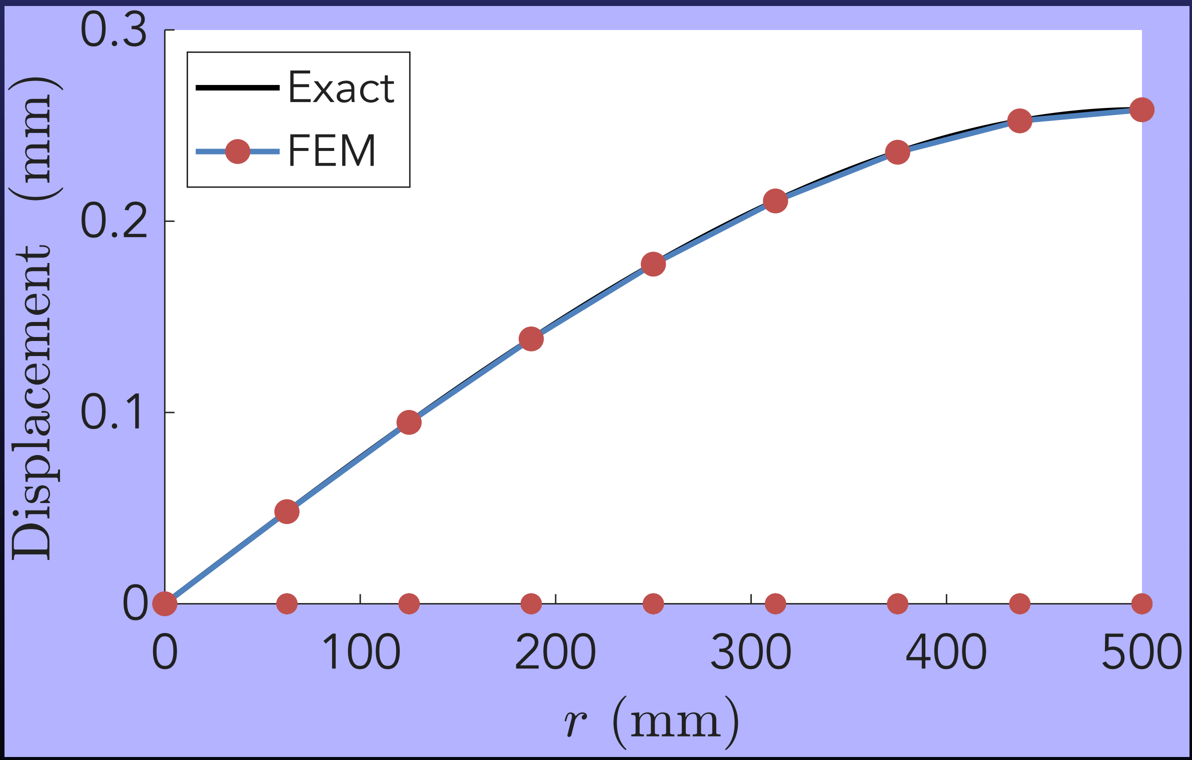


FEM Nodal Stress Smoothing

Eight Linear Elements

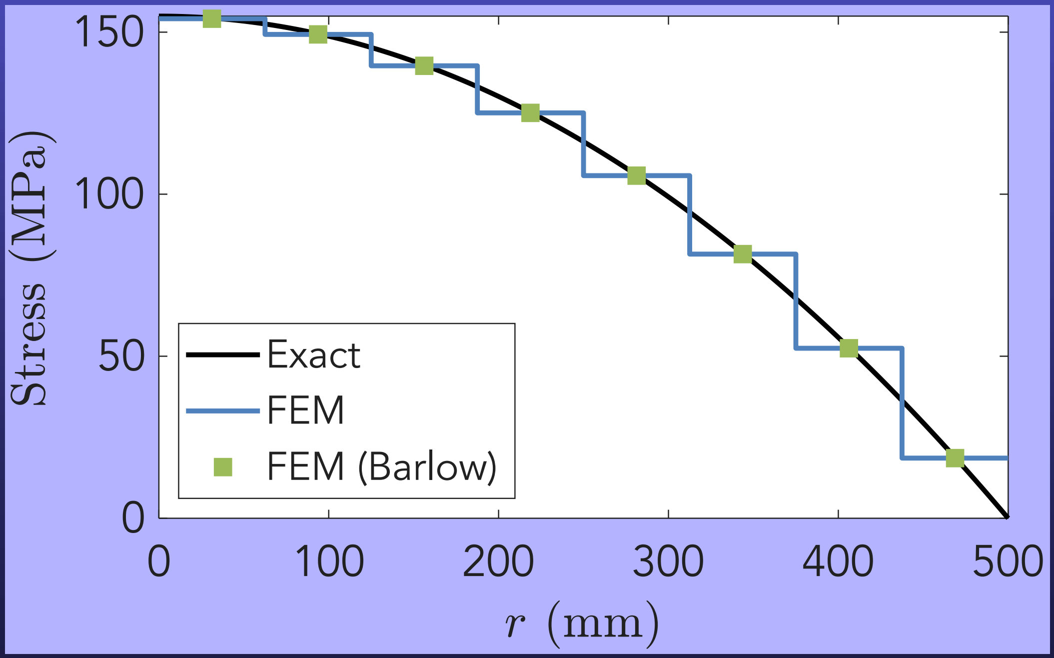
Consider a uniform mesh of eight two-node linear elements.

Displacement

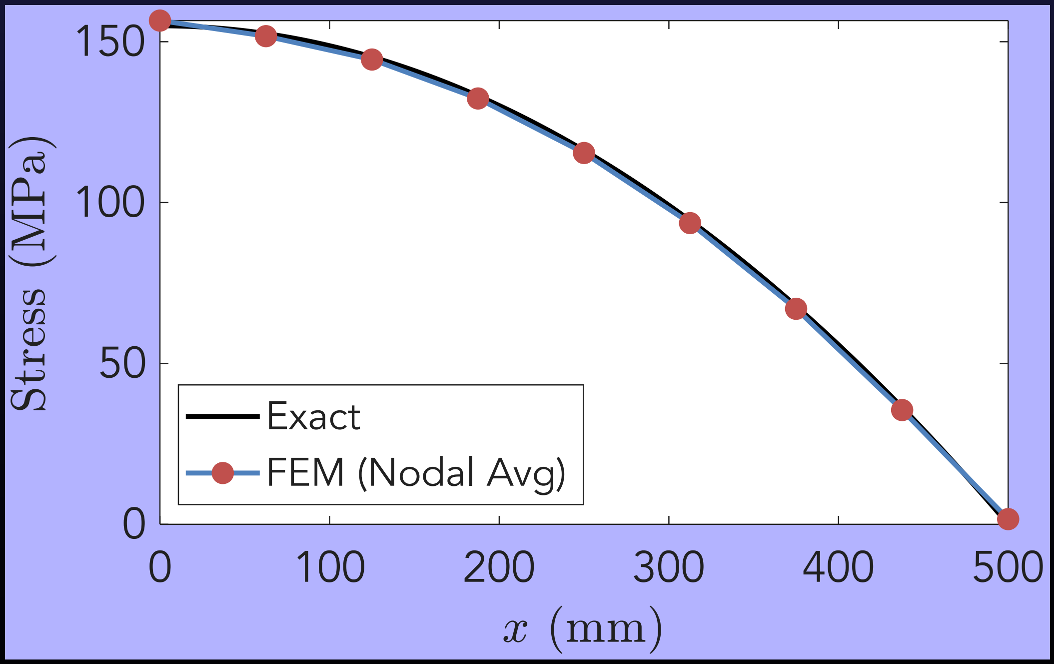


Piecewise Linear Displacement FEM approximation

Stress



Piecewise Constant Stress FEM approximation.



FEM Nodal Stress Smoothing

FEM Results Observations

1. As expected, the FEM stress approximation is less accurate than the displacement approximation.
2. The stress evaluated at the center midpoint equivalent to the one-point Gauss-Legendre Points (Barlow superconvergent points) is more accurate than other points within the element, especially at end nodes.
3. Improved stress approximation is obtained using smoothing. In this case, we used nodal interpolation of the average stress between connected elements. The nodes at the ends are extrapolated. Another approach is to use patch recovery for accurate nodal stress values.
4. The extrapolation at the last node tends to undershoot or overshoot with less accuracy than the interior interpolated points.
5. This simple averaging is not just for plotting; It gives a more accurate nodal stress than the constant values from adjacent elements at connected nodes.

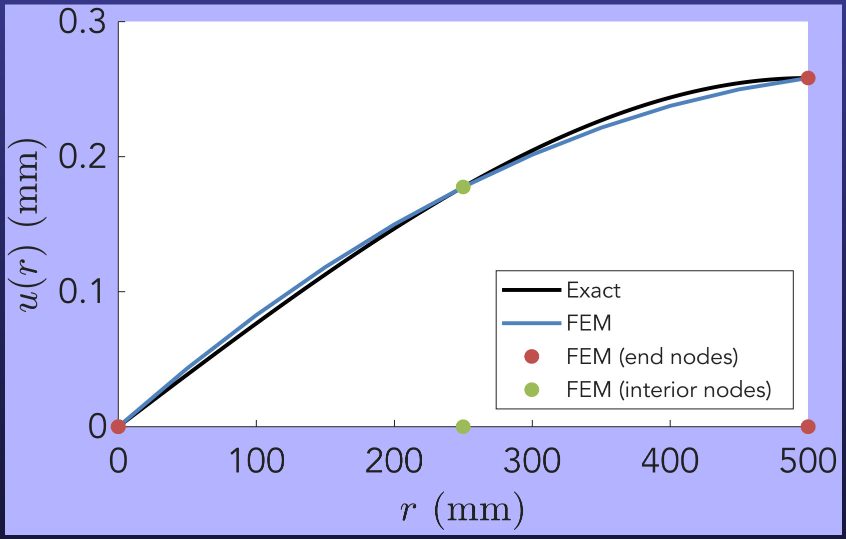
6. A generalization of this idea of interpolating the optimal Barlow stress points can be used with other stress post-processing methods, such as global interpolation and Superconvergent Patch Recovery (SPR) for accurate nodal stress values.
7. Since the rod is uniform with a constant cross-section area and material EA = constant, the FEM solution nodal displacements and midpoint stresses can be proven to be exact regardless of the body load, in this case, the linear varying Centrifugal force. **This is a special case not expected in general two and three-dimensional applications of FEA**. The phenomena of exact or higher accuracy than expected at other points is called **superconvergence**.
8. The FEM approximation converges to the exact analytical solution everywhere with mesh refinement.
9. Only a few elements are needed to match displacements everywhere.
10. Since the solution is smooth (high regularity), with a linear stress gradient, a uniform mesh with equally spaced nodes gave accurate solutions everywhere in the problem domain.

Uniform Mesh of Quadratic Elements

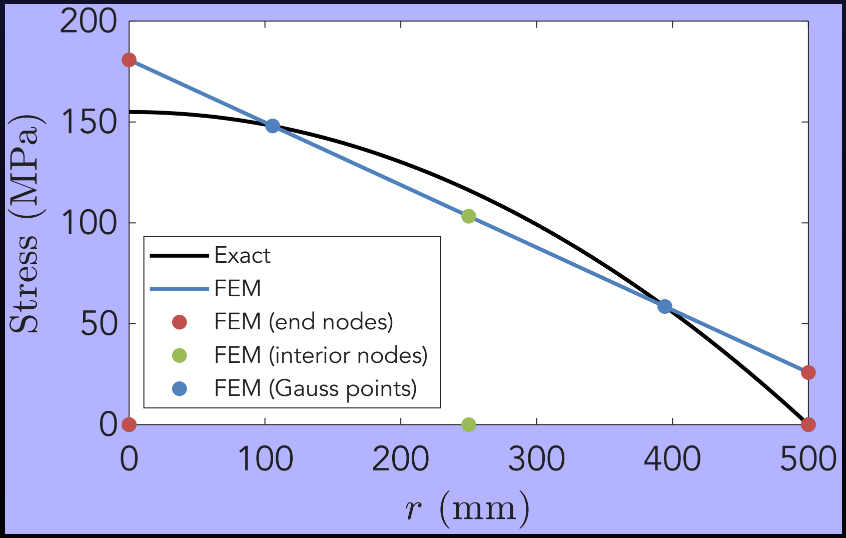
Let's do a mesh refinement study of 1, 2, and 4 quadratic elements with the same number of 3, 5, and 9 mesh nodes, respectively.

One Quadratic Element

Consider a mesh of a single 3-node quadratic element.



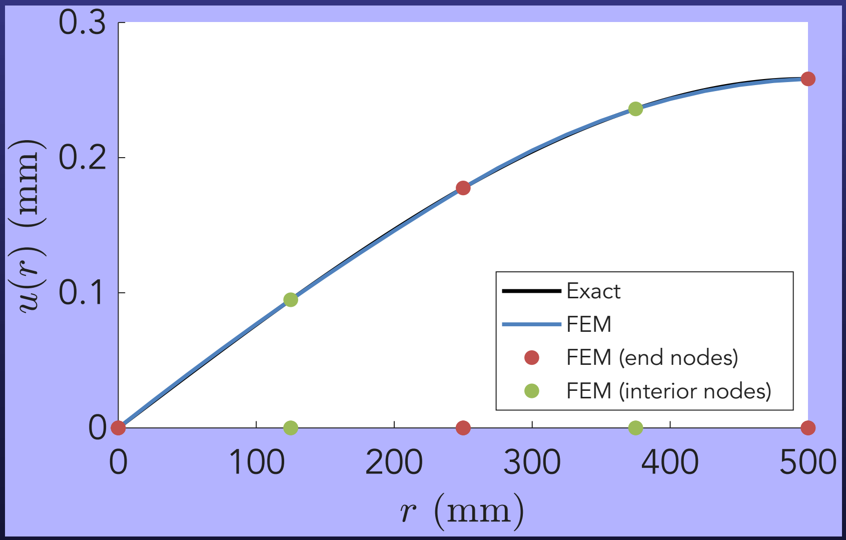
Quadratic Displacement FEM approximation



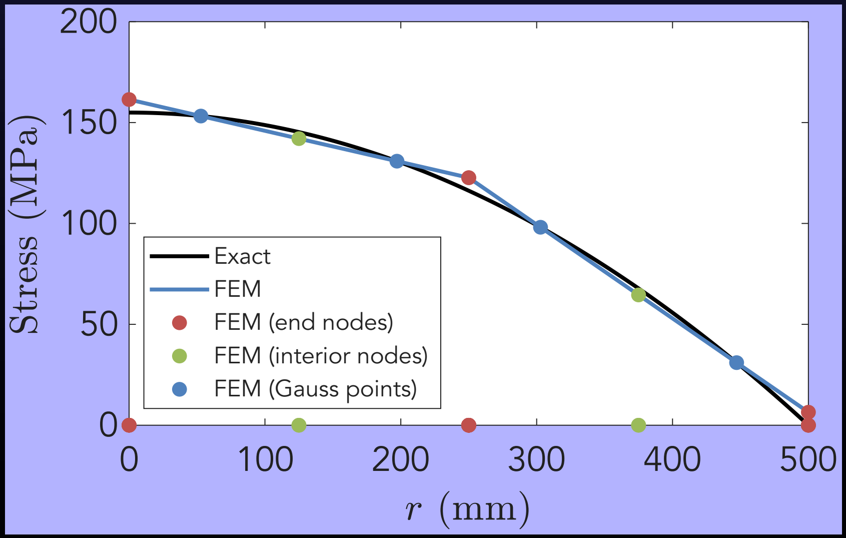
Linear Stress FEM approximation.

Two Quadratic Elements

Consider a uniform mesh of a relatively coarse mesh of two 3-node quadratic elements.



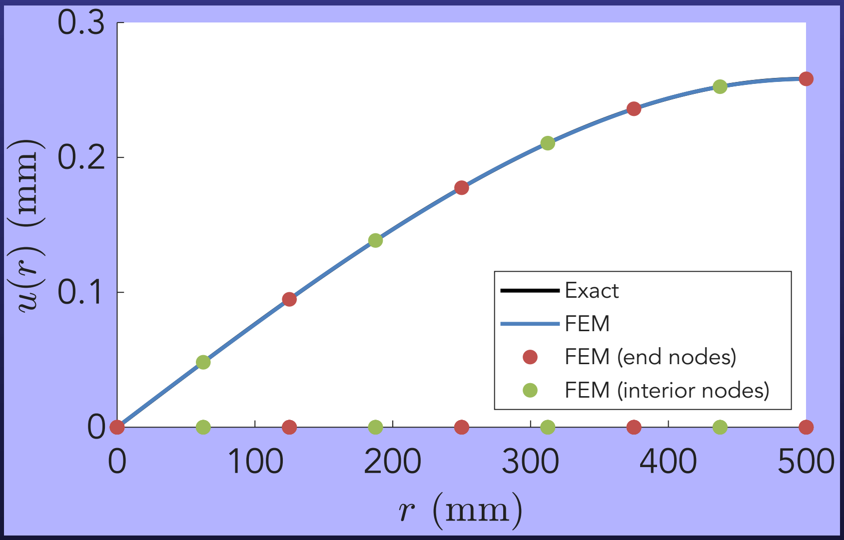
Piecewise Quadratic Displacement FEM approximation



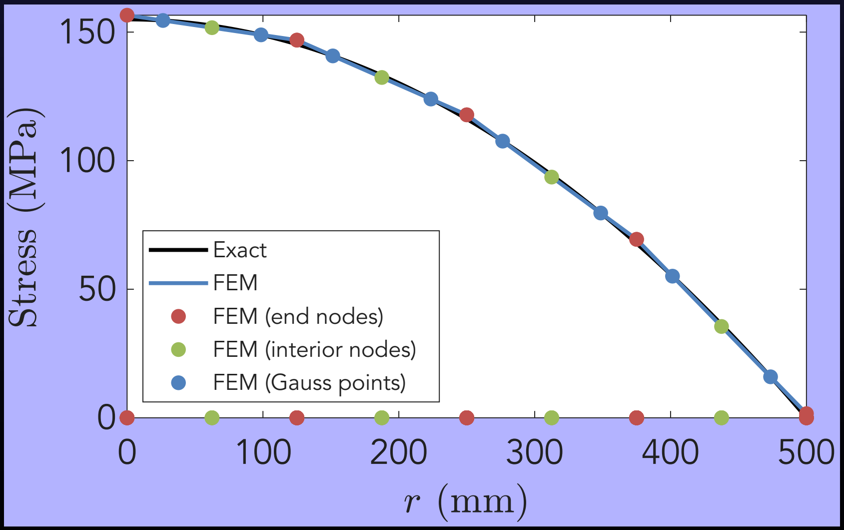
Piecewise Linear Stress FEM approximation.

Four Quadratic Elements

Consider a uniform mesh of four three-node quadratic elements.



Piecewise Quadratic Displacement FEM approximation



Piecewise Linear Stress FEM approximation.

Observations of results for Quadratic Element Meshes

1. Since the solution is smooth (high regularity), the quadratic element mesh solutions are more accurate than the linear mesh solutions for the same number of equations and unknowns.
2. The FEM piecewise constant stress approximation for 2-node linear displacement elements is less accurate than the piecewise linear approximation of the 3-node quadratic displacement elements.
3. The 3-node quadratic elements give the best stress approximation at the 2-point Gauss-Legendre Quadrature points as found by Barlow. The stress at these points can be used with a smoothing technique such as the superconvergent patch recovery (SPR) method to recover accurate stresses at nodes.
4. All commercial software such as Ansys, Abaqus, and Nastran use some sort of stress smoothing for nodal stresses (something that most user software user manuals do not explain, likely for competitive reasons).

Comparison of Stiffness Matrix Sparsity between 2-node linear and 3-node quadratic element meshes

1. Each row in the stiffness matrix represents force equilibrium at each node in the mesh.
2. For the linear element mesh, since the shape functions for the nodes connect with two adjacent 2-node elements, there are three nonzero coefficients per row.
3. For the quadratic element mesh, since the shape functions for connected nodes connect with two adjacent 3-node elements, there are five nonzero coefficients per row at these connected nodes.
4. For the quadratic mesh, since the shape function for the interior nodes is local to the element and not connected with adjacent elements, there are three nonzero coefficients per frow for these interior nodes.
5. There are more nonzero coefficients in the quadratic element mesh, requiring additional memory storage and compute time when solving equations.

Summary and Conclusions

- This detailed comparison between FEA and analytical solutions for a rotating elastic rod illustrates the practical considerations of mesh density and element order in achieving accurate results.
- The comparison between refined finite element meshes highlights the importance of mesh density and element order in achieving accurate stress and displacement predictions. It's fascinating to see how nodal interpolation (stress smoothing) can significantly enhance accuracy.
- The insights on nodal interpolation techniques and their impact on stress accuracy provide valuable guidance for optimizing FEA models.

Further observations

- For this model problem studied, since the analytical solution for the linear Centrifugal force has quadratic stress and cubic displacement, a single hierarchical basis p-version finite element or a global Rayleigh-Ritz basis approximation for displacement with a 3rd-order polynomial would capture the displacement and stress exactly everywhere.
- However, the point of this study was to examine mesh refinement of standard 2-node linear elements with standard Lagrange basis shape functions used in Commercial FEA software to give insights into the use of these types of elements in multi-dimensional complex problems.
- The global basis Rayleigh-Ritz method can be difficult to apply to complex geometries and does not provide a framework for general-purpose software. For these and other compelling reasons, the Rayleigh-Ritz method is not used in practice in industry except in special cases.

- Analytical solutions are generally available for simple shapes, such as one-dimensional, two-dimensional, and three-dimensional problems with separable coordinates.
- There are fewer nonlinear solutions to compare to, and in general, they are not available for complex non-linear problems. Nevertheless, a great deal can be gained by studying how well finite element solutions behave with mesh density relative to analytical solutions where they exist.
- Of course, in practice, we should never solve the FEA solution with one mesh; mesh refinement and solution iterations are needed to determine mesh sensitivity and convergence (grid independence) up to an acceptable tolerance within constraints on computer memory and processing time.

Correction

The plots of stress are labeled as MPa;
the correct units are kPa.

References

1. Thomas J.R. Hughes, *The Finite Element Method: Linear Static and Dynamic Finite Element Analysis*, 1987. Republished by Dover in 2000.
2. G. Strang and G.J. Fix, *An Analysis of the Finite Element Method*. 1973.
3. J. Barlow, 'Optimal Stress Locations in Finite Element Models', International Journal for Numerical Methods in Engineering, 10 (1976), 243-251.

We love FEA

The Finite Element Method (FEM) blends elegance with utility, rewarding mastery with the power to solve real-world problems with FEA.

Please share this FEA solution with others so they can also benefit from its insights.

–Thanks!