

Both problems should be solved.

Problem 1:

Computational modeling of dental crowns

1. Introduction

In this project we compare three different configurations for dental crown restoration. Figure 1 shows the schematic of a dental restoration. The top and bottom layers resemble the properties of enamel and dentin, respectively. We investigate the effect of the connective layer that substitutes the dentin-enamel junction (DEJ) in real teeth. This layer transmits loads from enamel to dentin and the gradual change of material properties reduces stress concentrations.

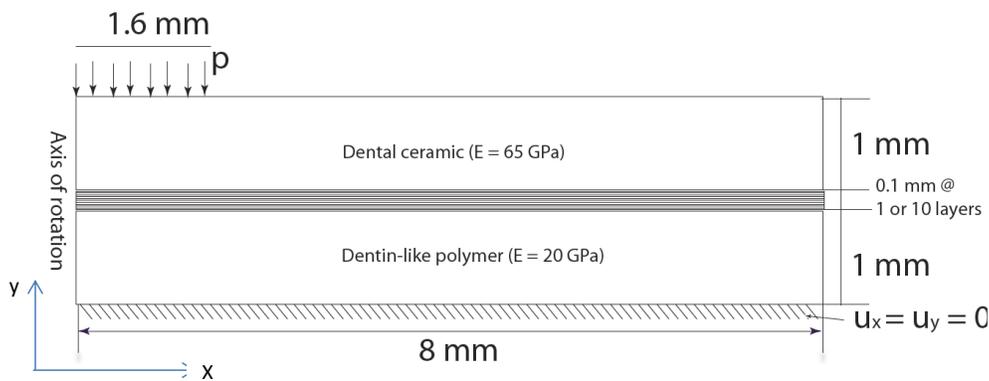


Figure 1. Schematic of dental restoration.

We analyze a cylinder with the axis of rotation (cylindrical symmetry) at $x = 0$. The dimensions, material properties, and boundary conditions are shown in figure 1. For simplicity we use Poisson ratio, $\nu = 0.3$ for all materials in our simulations. The boundary conditions are:

1. All displacements fixed at the bottom edge ($u_x = u_y = 0$)
2. Uniform pressure on $1/5^{\text{th}}$ of the top edge so that the resultant force is 120N. Make sure to take into account that the actual geometry is a cylinder so that the line load shown in the figure corresponds to a circle with uniform pressure p .
3. Symmetric boundary condition at $x = 0$ (this is the axis of rotation).

2. Problem cases

The three cases we study are governed by the following mappings of E from a dentin-like polymer side to dental ceramic:

- I. Linear variation of [48, 65).
- II. Dental Cement with constant elastic modulus of $E = 5 \text{ GPa}$.
- III. Linear variation of [20, 65)

Commented [JJ1]: What do the numbers in parenthesis mean?

The details of these maps are given in Table 1:

layer#	case I	case II	case III
1	48	5	24.091
2	49.7	5	28.182
3	51.4	5	32.273
4	53.1	5	36.364
5	54.8	5	40.455
6	56.5	5	44.545
7	58.2	5	48.636
8	59.9	5	52.727
9	61.6	5	56.818
10	63.3	5	60.909

Table 1: Variation of Elastic modulus for the connection between the top and bottom layers. The layer number is from the bottom layer.

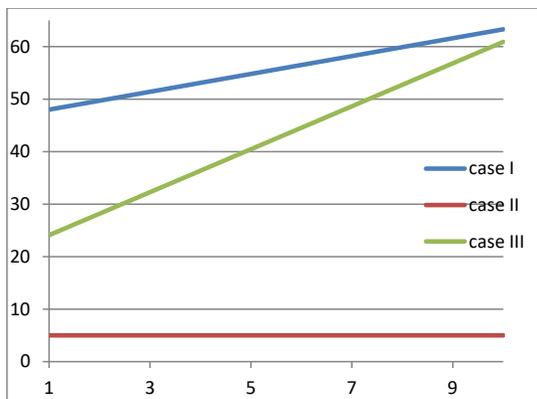


Figure 2: Distribution of Elastic modulus in the middle layer.

Figure 2 shows constant and linear curves that pass through the discrete values in Table 1. Two load cases I, and III represent Functionally-Graded Materials (FGM) where Elastic modulus changes through the thickness. Note that Elastic modulus takes one uniform value of each of the ten 0.01 mm layers based on the values given in Table 1. Case II corresponds to more conventional crowns where dentin- and enamel -like layers are simply connected by dental cement. Compared to base case I, case III maps to a lower elastic modulus at the bottom.

3. Finite Element Analysis (FEA) procedure

The steps needed for the complete finite element analysis of this problem are:

1. Geometry: We define top and bottom rectangles and 10 rectangles for the middle layers. We can use the same geometry for all three cases or use a simpler three layer representation for case II. Here are some important points:
 - a. The Keypoints of rectangles that coincide should be merged to ensure that the different rectangles are in fact connected during the simulation. This is done through:
Preprocessor -> Numbering Ctrl's -> Merge Items -> Choosing Keypoints and small enough TOLER (e.g. 1e-9)
 - b. The top line should be divided so that we can apply constant pressure to its left 1/5th segment. Use Preprocessor -> Modeling -> Operate -> Booleans -> Divide -> Line into 2 Ln's.
2. Material: We define 12 materials (for cases I and III) and 3 materials for case II based on the values given in figure 1 and Table 1.
3. Element type: We use the element type "Solid 8 node (183)" for all simulations. Make sure axisymmetric is chosen through the options provided for this element.
4. Meshing. Both steps of meshing can be done by "Mesh Tool":
 - a. Linking all areas with correct material properties.
 - b. Meshing all areas with the mesher.
5. Boundary conditions: Boundary conditions are shown in figure 1.
6. Solution. Since this is a linear static problem only one linear analysis is needed.
7. Postprocessing: This step is further discussed below.

4. Deliverables:

The final report should include the following items and files:

- 1) **(90 points)** In smart mesh tool use the finest mesh that Ansys can successfully solve the problem, or alternatively use smart mesh number 3. Note smart mesh size options correspond to 10 for the coarsest to 1 for the finest meshes generated. Provide: Contour plots of 1st principal stress for all 3 cases with the following specifications. For each load case show the nodal contour plots for the converged (step 1) 1st principal stress and with user-specified lower and upper bounds of -20 MPa and 10 MPa respectively. The contour plots should be given for: i) the entire domain, ii) the lower layer (dentin-like polymer), iii) mid layers (connection), and iv) the top layer (dental ceramic).

Refer to this video:

<https://rezaabedi.com/wp-content/uploads/Courses/FEM/choosingDifferentMaterials4Plot.mp4>

for a short overview of how to select and plot elements of only certain layers.

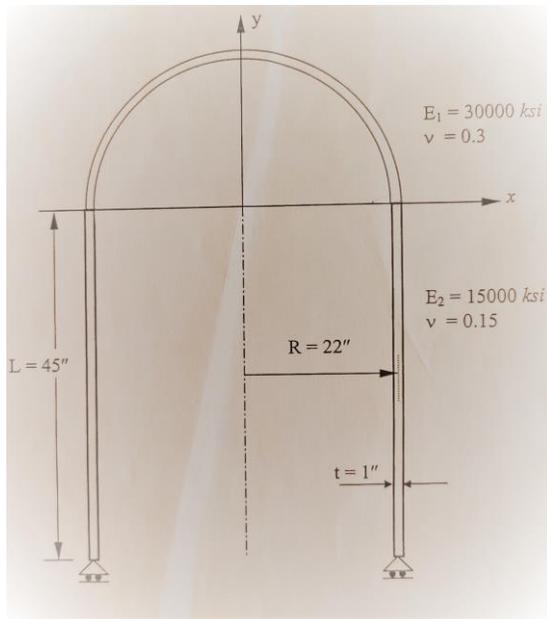
- 2) **(60 points)** Table of min/max of 1st principal stress for all simulations and all regions mentioned above. Table 2 provides a template for presenting the results:

	σ^1	Case I	Case II	Case III
All domain	min			
	max			
Bottom layer	min			
	max			
Mid layer(s)	min			
	max			
Top layer	min			
	max			

Table 2: Template for presenting max/min principal stress, σ^1 , for all cases.

- 3) **(25 points)** Plots of both minimum and maximum of σ^1 against case number for i) all domain, ii) bottom layer, iii) mid layer(s), and iv) top layer.
- 4) **(25 points)** Discuss the differences between different cases (answer in **less than 2 sentences** for each question):
- Fracture often initiates due to high tensile stresses. Discuss how different cases compare in terms of their maximum tensile stress (maximum σ^1) within each of the layers as well as for the entire domain.
 - Which of the layers experience the highest stresses?
 - Does the FGM design of case I perform better than case II?

Problem 2: Pressure Vessel (source: Prof. Glaucio Paulino, GTech)



An axisymmetric pressure vessel is constructed from a cylindrical tube with hemispherical head cap and is subjected to an internal pressure loading of 1000 psi . The cylindrical portion and the hemispherical portion are made of two different materials. The structure, loading, material properties and supports are all symmetric. The centerline radius is shown in the figure.

Estimate the displacements at the critical points (connection points inside and outside the vessel and the top point of the hemispherical portion and any other point that your stress contour plots suggest to be critical) of the pressure vessel, axial stress, circumferential stress at both inside and outside surface. **Provide contour plots of σ_1 (1st principal), σ_{xx} , σ_{yy} , σ_{zz} , and u_{sum} .**

Note: 1) Only make the geometry and mesh the right side of the domain as this is a problem with axisymmetry along the y axis (same as the dental crown project);

2) You can still use 8 node solid elements (recommended), but ensure that there are at least two to four elements along the thickness using smart mesh size option. You can also consider using the shell element.

(150 points)