

Computational modeling of dental crowns

1. Introduction

In this project we compare four 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.

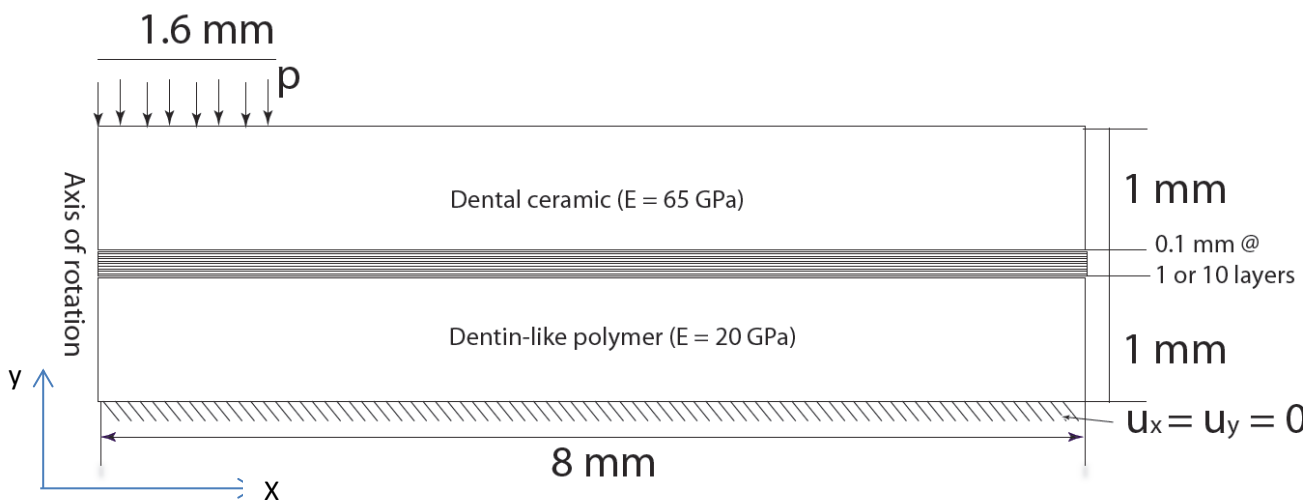


Figure 1. Schematic of dental restoration.

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 1/5th 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 four 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)

IV. Quadratic variation of [48, 65).

The details of these maps are given in Table 1:

layer#	case I	case II	case III	case IV
1	48	5	24.091	48
2	49.7	5	28.182	51.32
3	51.4	5	32.273	54.28
4	53.1	5	36.364	56.88
5	54.8	5	40.455	59.12
6	56.5	5	44.545	61
7	58.2	5	48.636	62.52
8	59.9	5	52.727	63.68
9	61.6	5	56.818	64.48
10	63.3	5	60.909	64.92

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

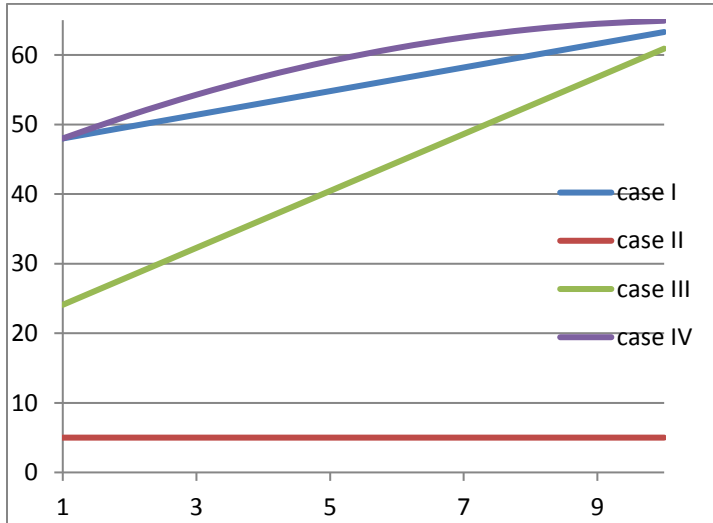


Figure 2: Distribution of Elastic modulus in the middle layer.

Figure 2 shows the linear and quadratic curves that pass through the discrete values in Table 1. Three load cases I, III, and IV 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 while load IV takes a quadratic variation.

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 four 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 -> NumberingCtrls -> 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, III, and IV) 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) **(30 points)** Convergence studies. The finite element solutions are not exact and special care should be taken in the interpretation of the results. Principal stresses are the values we compare in the following steps and we want them to be accurate enough. To achieve this, simulate case III with smart mesh refinement levels of 5, 4, 3, 2, 1 (medium to very fine mesh). If your computer does not permit simulating very fine meshes, stop at the finest level that you can simulate. By inspecting stress contour plots of 1st principal stress, form a table that lists minimum and maximum values (two columns), of principal stress of each simulation from the coarsest to the finest (each simulation's 2 numbers in a separate row). Comment on the convergence of these values as finer meshes are used.
- 2) **(60 points)** For each of the four cases show the nodal contour plots for the converged (step 1) 1st principal stress on undeformed geometry, without element boundaries, 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).

- 3) **(60 points)** Table of min/max of 1st principal stress for all simulations and all four regions mentioned above. Table 2 provides a template for presenting the results:

	σ^1	Case I	Case II	Case III	Case IV
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.

- 4) **(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.
- 5) **(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 performs better than case II?
 - Compare the two linear FGM designs of case I and III.