

## \Large ASSIGNMENT: 2

Name : HARSHIT SINGH  
 ROLL NO. : 22CE01012  
 GITHUB ID: sengarharshit

### QUESTION 1

**Problem Statement:** Deformation of a Square Body

Given the two-dimensional body defined by the region:

$$\mathcal{B} = \{(X_1, X_2) \mid 0.1 < X_1 < 1, \quad 0.1 < X_2 < 1\},$$

and the displacement field:

$$u_1 = \mathbf{u} \cdot \mathbf{e}_1 = 0.2 \ln(1 + X_1 + X_2), \quad u_2 = \mathbf{u} \cdot \mathbf{e}_2 = 0.2 \exp X_1.$$

**Objective:** Plot the displaced shape of the body. We proceed by defining a set of characteristic lines (a grid) on the reference body. For every point  $(X_1, X_2)$  on these lines, we compute the deformed position  $(x_1, x_2)$  using:

$$\begin{aligned} x_i &= X_i + u_i \\ x_1 &= X_1 + 0.2 \ln(1 + X_1 + X_2) \\ x_2 &= X_2 + 0.2 \exp X_1 \end{aligned}$$

using Plots

```
# Function to calculate deformed coordinates
function get_deformed_coords(X1, X2)
    # Displacement field definitions
    u1 = 0.2 * log(1 + X1 + X2)
    u2 = 0.2 * exp(X1)

    # New coordinates (x = X + u)
    x1_new = X1 + u1
    x2_new = X2 + u2
    return x1_new, x2_new
end

# Initialize the plot
p = plot(aspect_ratio=:equal, legend=:topleft, title="Deformation of Characteri
```

```

# --- Plot Horizontal Characteristic Lines (Constant X2) ---
# We take fixed values of X2 and vary X1
X2_fixed_vals = range(0.1, 1, length=10) # 10 horizontal lines
X1_continuous = range(0.1, 1, length=100) # High resolution for smooth curves

for X2_const in X2_fixed_vals
    # Original Line Arrays
    orig_x = [val for val in X1_continuous]
    orig_y = [X2_const for val in X1_continuous]

    # Deformed Line Arrays
    def_x = Float64[]
    def_y = Float64[]

    for X1_val in X1_continuous
        dx, dy = get_deformed_coords(X1_val, X2_const)
        push!(def_x, dx)
        push!(def_y, dy)
    end

    # Plot (only add label for the first line to keep legend clean)
    label_orig = (X2_const == X2_fixed_vals[1]) ? "Original Mesh" : ""
    label_def = (X2_const == X2_fixed_vals[1]) ? "Deformed Mesh" : ""

    plot!(p, orig_x, orig_y, color=:blue, linestyle=:dash, label=label_orig, alpha=0.5)
    plot!(p, def_x, def_y, color=:red, linewidth=1.5, label=label_def)
end

# --- Plot Vertical Characteristic Lines (Constant X1) ---
# We take fixed values of X1 and vary X2
X1_fixed_vals = range(0.1, 1, length=10) # 10 vertical lines
X2_continuous = range(0.1, 1, length=100)

for X1_const in X1_fixed_vals
    # Original Line Arrays
    orig_x = [X1_const for val in X2_continuous]
    orig_y = [val for val in X2_continuous]

    # Deformed Line Arrays
    def_x = Float64[]
    def_y = Float64[]

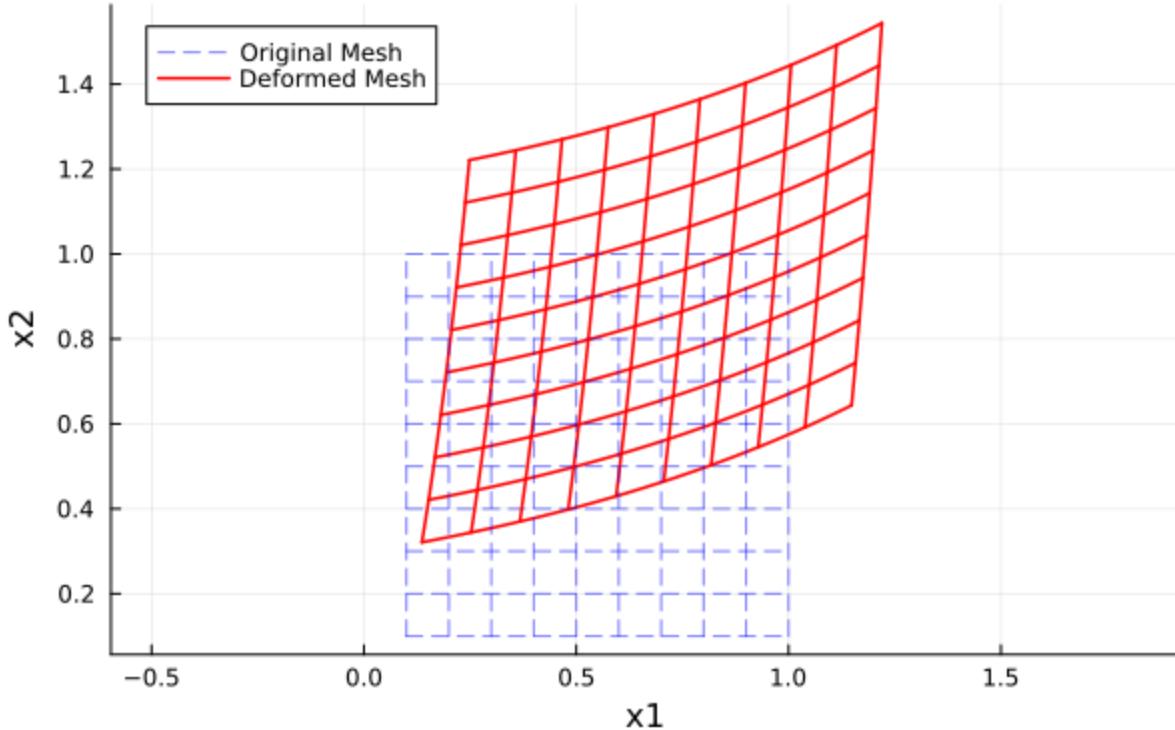
    for X2_val in X2_continuous
        dx, dy = get_deformed_coords(X1_const, X2_val)
        push!(def_x, dx)
        push!(def_y, dy)
    end

    plot!(p, orig_x, orig_y, color=:blue, linestyle=:dash, label="", alpha=0.5)
    plot!(p, def_x, def_y, color=:red, linewidth=1.5, label="")
end

```

```
# Display the final plot
display(p)
```

## Deformation of Characteristic Lines



## QUESTION 2

### Problem Statement: Annular Body Deformation

Consider a two-dimensional annular body  $\mathcal{B}$  with respect to a rectangular Cartesian coordinate system with orthonormal vectors  $(\mathbf{e}_1, \mathbf{e}_2)$ . Let the coordinates of a point be denoted by  $(X_1, X_2)$ . The body occupies the region:

$$\mathcal{B} = \{(X_1, X_2) \mid 1 < \sqrt{X_1^2 + X_2^2} < 2\}.$$

Due to the annular geometry, we utilize a polar coordinate system  $(R, \theta)$ , related to the rectangular coordinates by:

$$R = \sqrt{X_1^2 + X_2^2} \quad \text{and} \quad \theta = \tan^{-1}(X_2/X_1).$$

The orthonormal base vectors  $(\mathbf{e}_r, \mathbf{e}_\theta)$  of the polar coordinate system are related to  $(\mathbf{e}_1, \mathbf{e}_2)$  by the transformation:

$$\mathbf{e}_r = \cos \theta \mathbf{e}_1 + \sin \theta \mathbf{e}_2, \quad \mathbf{e}_\theta = -\sin \theta \mathbf{e}_1 + \cos \theta \mathbf{e}_2.$$

We examine the deformed configuration of this annular body under the following deformation field:

$$u_r = \mathbf{u} \cdot \mathbf{e}_r = 0.4(R - 1)^2 \cos 3\theta$$

$$u_\theta = \mathbf{u} \cdot \mathbf{e}_\theta = 0.4(R - 1)^3$$

```

using Plots
R_vals = range(1, 2, length=20)
Theta_vals = range(0, 2*pi, length=100)

# Initialize arrays to store coordinates
# X_orig, Y_orig: Original coordinates
# X_def, Y_def: Deformed coordinates
X_orig = Float64[]
Y_orig = Float64[]
X_def = Float64[]
Y_def = Float64[]

# 2. Loop through the domain to calculate displacements
for r in R_vals
    for theta in Theta_vals
        # --- Original Configuration ---
        # Convert Polar (R, Theta) to Cartesian (X1, X2)
        x1 = r * cos(theta)
        x2 = r * sin(theta)

        push!(X_orig, x1)
        push!(Y_orig, x2)

        # --- Calculate Displacement Field ---
        # Given formulas:
        # u_r = 0.4 * (R - 1)^2 * cos(3*theta)
        # u_theta = 0.4 * (R - 1)^3
        u_r = 0.4 * (r - 1)^2 * cos(3*theta)
        u_theta = 0.4 * (r - 1)^3

        # --- Convert Displacement to Cartesian Components ---
        # The displacement vector u = u_r * e_r + u_theta * e_theta
        # We need u_1 and u_2 (components along e1 and e2)
        # u_1 = u_r * cos(theta) - u_theta * sin(theta)
        # u_2 = u_r * sin(theta) + u_theta * cos(theta)

        u1 = u_r * cos(theta) - u_theta * sin(theta)
        u2 = u_r * sin(theta) + u_theta * cos(theta)

        # --- Deformed Configuration ---
        # x_new = x_old + displacement
        x1_new = x1 + u1
        x2_new = x2 + u2
    end
end

```

```

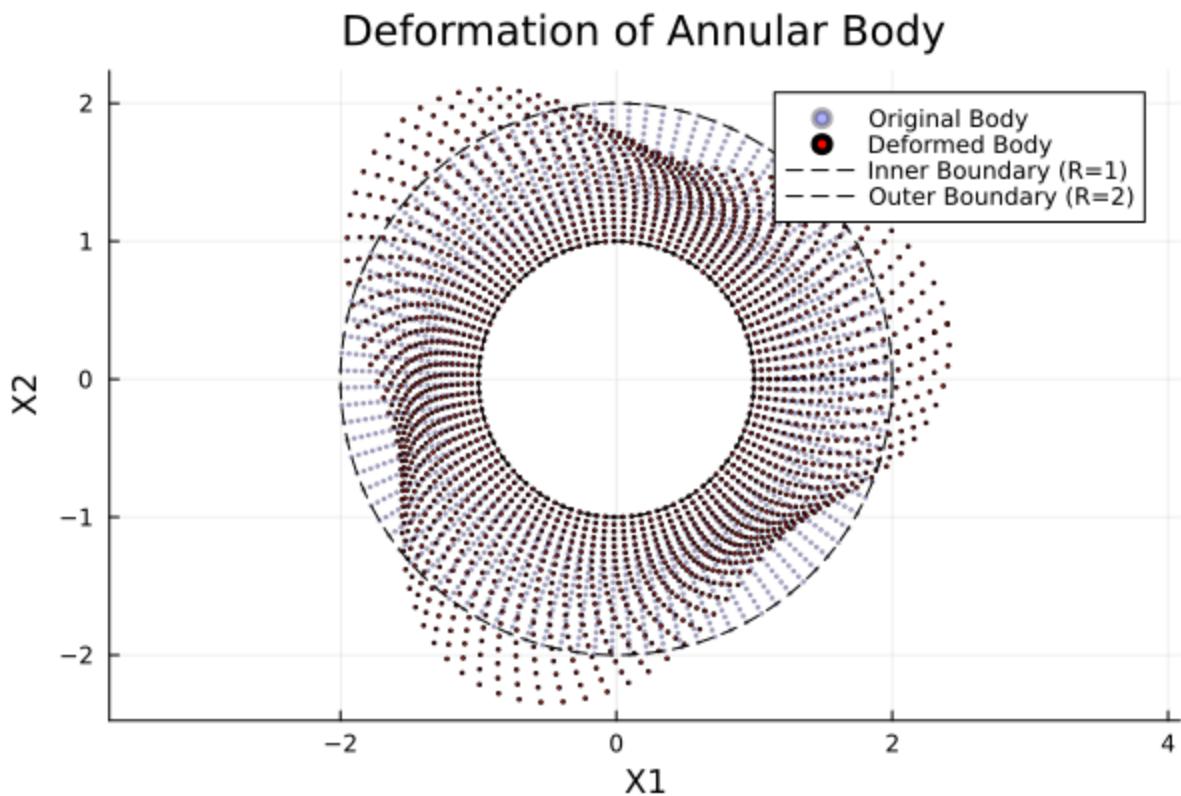
        push!(X_def, x1_new)
        push!(Y_def, x2_new)
    end
end

# 3. Plotting
# We plot the points as a scatter plot for visualization clarity
scatter(X_orig, Y_orig, label="Original Body", color=:blue, markersize=1, alpha=.5)
scatter!(X_def, Y_def, label="Deformed Body", color=:red, markersize=1, title="Deformation of Annular Body")

# Add boundary circles for reference (optional visual aid)
theta_circle = range(0, 2*pi, length=200)
plot!(1 .* cos.(theta_circle), 1 .* sin.(theta_circle), line=:dash, color=:black)
plot!(2 .* cos.(theta_circle), 2 .* sin.(theta_circle), line=:dash, color=:black)

xlabel!("X1")
ylabel!("X2")

```



## QUESTION 3

### Problem Statement: Mixed Coordinate Deformation

Consider a two-dimensional body  $\mathcal{B}$  defined in a rectangular Cartesian coordinate system:

$$\mathcal{B} = \{(X_1, X_2) \mid 0.1 < X_1 < 1, \quad 0.1 < X_2 < 1\},$$

subjected to a displacement field where the components are given in the polar basis ( $\mathbf{e}_r, \mathbf{e}_\theta$ ), but the magnitudes depend on the Cartesian coordinates ( $X_1, X_2$ ):

$$u_r = \mathbf{u} \cdot \mathbf{e}_r = 0.2 \exp X_1$$

$$u_\theta = \mathbf{u} \cdot \mathbf{e}_\theta = 0.2 \ln(1 + X_1 + X_2).$$

To plot the deformed shape in the Cartesian plane, we utilize the transformation between the polar basis and the Cartesian basis, where  $\theta = \tan^{-1}(X_2/X_1)$ :

$$u_1 = u_r \cos \theta - u_\theta \sin \theta$$

$$u_2 = u_r \sin \theta + u_\theta \cos \theta$$

```
using Plots

# 1. Define the Domain
# The body is a square region from 0.1 to 1 in both X1 and X2
X_vals = range(0.1, 1, length=20)
Y_vals = range(0.1, 1, length=20)

# Initialize arrays to store coordinates
X_orig = Float64[]
Y_orig = Float64[]
X_def = Float64[]
Y_def = Float64[]

# 2. Iterate through the grid
for x1 in X_vals
    for x2 in Y_vals
        # --- Store Original Configuration ---
        push!(X_orig, x1)
        push!(Y_orig, x2)

        # --- Calculate Displacement Field ---
        # First, determine the polar angle for the current Cartesian point
        # atan(y, x) computes the correct quadrant angle (theta)
        theta = atan(x2, x1)

        # Calculate the polar displacement components given in the problem
        u_r = 0.2 * exp(x1)
        u_theta = 0.2 * log(1 + x1 + x2)

        # --- Transform to Cartesian Displacement Components ---
        # u1 = u_r * cos(theta) - u_theta * sin(theta)
        # u2 = u_r * sin(theta) + u_theta * cos(theta)
        u1 = u_r * cos(theta) - u_theta * sin(theta)
        u2 = u_r * sin(theta) + u_theta * cos(theta)

        # --- Calculate Deformed Configuration ---
        x1_new = x1 + u1
```

```
x2_new = x2 + u2

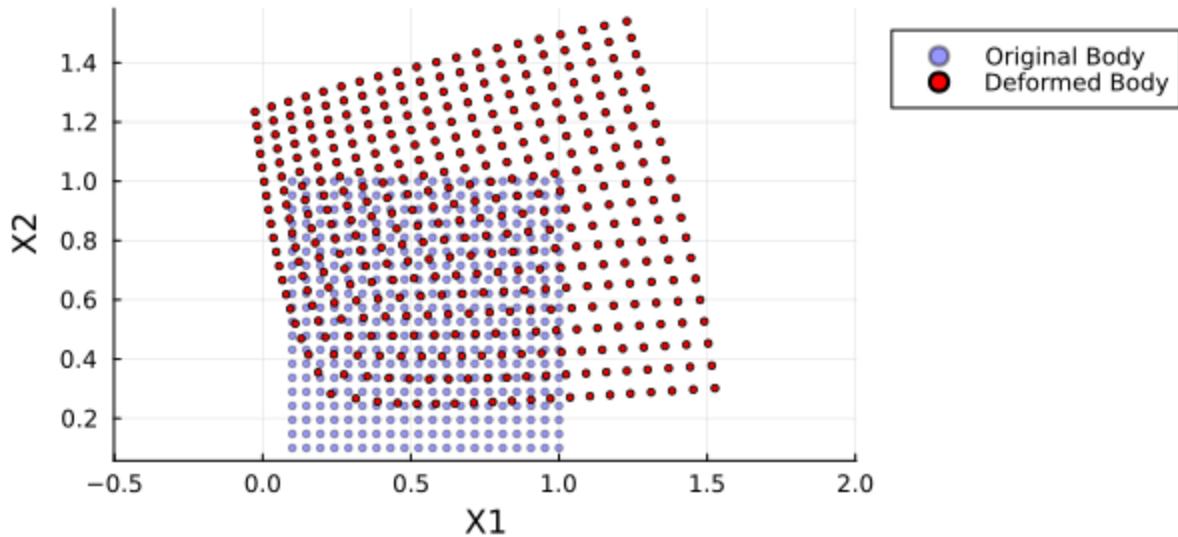
push!(X_def, x1_new)
push!(Y_def, x2_new)
end
end

# 3. Plotting
# Plot Original Shape
scatter(X_orig, Y_orig,
       label="Original Body",
       color=:blue,
       markersize=2,
       alpha=0.4,
       aspect_ratio=:equal,
       legend=:outertopright
)

# Plot Deformed Shape
scatter!(X_def, Y_def,
       label="Deformed Body",
       color=:red,
       markersize=2,
       title="Mixed Coordinate Deformation"
)

# Add axis labels
xlabel!("X1")
ylabel!("X2")
```

## Mixed Coordinate Deformation



### QUESTION 4

#### Finite Element Analysis of a Rectangular Plate with a Circular Hole

##### 1. Problem Definition

We model the stress distribution in a steel rectangular plate with a central circular hole under uniaxial tension.

##### Geometric and Material Parameters:

- **Dimensions:** Length  $L = 1000$  mm, Height  $H = 400$  mm.
- **Defect:** Central circular hole with radius  $R = 75$  mm.
- **Thickness:**  $t = 2$  mm.
- **Young's Modulus:**  $E = 210$  GPa =  $2.1 \times 10^5$  MPa.
- **Poisson's Ratio:**  $\nu = 0.3$ .
- **Loading:** Distributed tensile load  $q = 100$  N/mm along the right edge.

**Load Conversion:** Since the analysis is 2D plane stress, we convert the line load  $q$  into a traction (stress) applied to the edge surface:

$$T_x = \frac{q}{t} = \frac{100 \text{ N/mm}}{2 \text{ mm}} = 50 \text{ MPa.}$$

## 2. Governing Partial Differential Equations (Linear Elasticity)

The problem is governed by the equations of linear elastostatics. We seek the displacement field  $\mathbf{u}$  such that:

1. **Equilibrium Equation** (neglecting body forces):

$$\nabla \cdot \boldsymbol{\sigma} = \mathbf{0} \quad \text{in } \Omega$$

2. **Kinematic Equation** (Strain-Displacement):

$$\boldsymbol{\epsilon} = \frac{1}{2}(\nabla \mathbf{u} + (\nabla \mathbf{u})^T)$$

3. **Constitutive Equation** (Hooke's Law for Isotropic Material):

$$\boldsymbol{\sigma} = \mathbb{C} : \boldsymbol{\epsilon} \implies \boldsymbol{\sigma} = \lambda(\text{tr } \boldsymbol{\epsilon})\mathbf{I} + 2\mu\boldsymbol{\epsilon}$$

where  $\lambda$  and  $\mu$  are the Lamé parameters derived from  $E$  and  $\nu$ .

4. **Boundary Conditions:**

- **Dirichlet (Fixed):**  $\mathbf{u} = \mathbf{0}$  on the left edge (or symmetry conditions if modeling half-plate).
- **Neumann (Traction):**  $\boldsymbol{\sigma} \cdot \mathbf{n} = \mathbf{T}$  on the right edge, where  $\mathbf{T} = (50, 0)$  MPa.
- **Free Surface:**  $\boldsymbol{\sigma} \cdot \mathbf{n} = \mathbf{0}$  on the hole boundary and top/bottom edges.

## 3. Finite Element Analysis Steps in Abaqus

### 1. Part Module (Geometry Definition)

- **Create Part:** Select 2D Planar, Deformable, Shell.
- **Sketch Geometry:**
  - Draw a rectangle with corners at  $(0, 0)$  and  $(1000, 400)$ .
  - Draw a circle at the center coordinates  $(500, 200)$  with a radius  $R = 75$  mm.
  - **Name:** Plate\_Hole.

### 2. Property Module (Material Definition)

- **Create Material:** Name it Steel.
  - **Mechanical → Elasticity → Elastic:**
  - Young's Modulus  $E = 210000$  MPa (consistent units are crucial).
  - Poisson's Ratio  $\nu = 0.3$ .
- **Create Section: Solid, Homogeneous.**

- Plane Stress/Strain thickness  $t = 2$  mm.
- **Assign Section:** Select the entire plate surface and assign the created section.

### 3. Assembly Module

- **Create Instance:** Select `Plate_Hole`. Choose **Dependent** (mesh on part).

### 4. Step Module

- **Create Step: Static, General.**
- **Nlgeom:** set to **Off** (Linear elastic analysis).
- **Time period:** 1.0 (default).

### 5. Load Module (Boundary Conditions & Loads)

- **Boundary Condition (BC-1):**
  - Type: **Displacement/Rotation**.
  - Select the **Left Edge ( $X = 0$ )**.
  - Set  $U1 = U2 = UR3 = 0$  (Fixed support).
- **Load (Load-1):**
  - Type: **Surface Traction or Shell Edge Load**.
  - Select the **Right Edge ( $X = 1000$ )**.
  - **Magnitude:**
    - If using **Shell Edge Load**: Input 100 N/mm.
    - If using **Surface Traction**: Input stress  $\sigma = 50$  MPa.
  - **Direction:** Normal vector outwards (Tensile).

### 6. Mesh Module

- **Seeding:**
  - **Global Seeds:** Approximate size 20 – 30 mm.
  - **Local Seeds:** Select the circular hole edge and seed with a finer size (e.g., 5 mm) to capture the stress concentration factor  $K_t$ .
- **Element Type:**
  - Family: **Plane Stress**.
  - Geometric Order: **Linear** (CPS4R) or **Quadratic** (CPS8R) for better accuracy.
- **Mesh Part:** Click **Mesh Part** to generate the grid.

### 7. Job Module

- **Create Job:** Name `Job-Plate-Hole`.
- **Submit:** Click Submit and wait for `Completed` status.

## 8. Visualization (Post-Processing)

- **Open ODB:** Open the results file.
- **Plot Contours:** Select **Plot Contours on Deformed Shape**.
- **Field Output:** Select **S (Mises)** for Von Mises Stress or **S11** for horizontal stress.
- **Check Max Stress:** Inspect the top and bottom of the hole. Theoretical Max Stress should be roughly:

$$\sigma_{max} \approx 3 \times \sigma_{nom} = 3 \times 50 \text{ MPa} = 150 \text{ MPa}$$

```

using Gridap
using GridapGmsh
import GridapGmsh: gmsh

# =====
# 1. ROBUST GMSH INITIALIZATION
# =====
# This block prevents the "Already initialized" error if you re-run the cell.
if GridapGmsh.gmsh.isInitialized() == 1
    GridapGmsh.gmsh.finalize()
end
GridapGmsh.gmsh.initialize()
gmsh.clear()
gmsh.model.add("Plate_Hole_2D")

# =====
# 2. GEOMETRY & MESH (Bottom-Up Approach)
# =====
L = 1000.0
H = 400.0
R = 75.0
h = 20.0 # Element size (User variable 'h' defined here)

# -- Points --
p1 = gmsh.model.geo.addPoint(0, 0, 0, h)
p2 = gmsh.model.geo.addPoint(L, 0, 0, h)
p3 = gmsh.model.geo.addPoint(L, H, 0, h)
p4 = gmsh.model.geo.addPoint(0, H, 0, h)

# Center and Circle Points
p5 = gmsh.model.geo.addPoint(L/2, H/2, 0, h)
p6 = gmsh.model.geo.addPoint((L/2)+R, H/2, 0, h)
p7 = gmsh.model.geo.addPoint((L/2)-R, H/2, 0, h)
p8 = gmsh.model.geo.addPoint((L/2), (H/2)+R, 0, h)
p9 = gmsh.model.geo.addPoint((L/2), (H/2)-R, 0, h)

# -- Lines & Arcs --
l1 = gmsh.model.geo.addLine(p1, p2)
l2 = gmsh.model.geo.addLine(p2, p3)
l3 = gmsh.model.geo.addLine(p3, p4)

```

```

14 = gmsh.model.geo.addLine(p4, p1)

# Note: Arcs must form a closed loop.
# p8(top) -> p6(right) -> p9(bottom) -> p7(left) -> p8(top) is a valid loop.
15 = gmsh.model.geo.addCircleArc(p8, p5, p6)
16 = gmsh.model.geo.addCircleArc(p6, p5, p9)
17 = gmsh.model.geo.addCircleArc(p9, p5, p7)
18 = gmsh.model.geo.addCircleArc(p7, p5, p8)

# -- Loops & Surface --
plate_loop = gmsh.model.geo.addCurveLoop([11, 12, 13, 14])
hole_loop = gmsh.model.geo.addCurveLoop([15, 16, 17, 18])
surface = gmsh.model.geo.addPlaneSurface([plate_loop, hole_loop])

gmsh.model.geo.synchronize()

# -- Physical Groups (CRITICAL FOR GRIDAP) --
# We assume 14 is Left (x=0) and 12 is Right (x=L) based on point order.
gmsh.model.addPhysicalGroup(1, [14], 1, "LeftBoundary")
gmsh.model.addPhysicalGroup(1, [12], 2, "RightBoundary")
gmsh.model.addPhysicalGroup(2, [surface], 3, "PlateSurface")

# -- Generate and Write --
gmsh.model.mesh.generate(2)
msh_file = "plate_hole_robust.msh"
gmsh.write(msh_file)
GridapGmsh.gmsh.finalize()
println("Mesh saved to $msh_file")

# =====
# 3. FINITE ELEMENT ANALYSIS
# =====

# Load Mesh from file (Bypasses tagging errors)
model = GmshDiscreteModel(msh_file)

# FE Spaces
order = 1
reffe = ReferenceFE(lagrangian, VectorValue{2,Float64}, order)
V0 = TestFESpace(model, reffe; dirichlet_tags=["LeftBoundary"])
U = TrialFESpace(V0, VectorValue(0.0,0.0))

# Material Params
thickness = 2.0
traction_val = 100.0 / thickness # 50 MPa
E = 2.1e5
v = 0.3
λ = (E*v)/((1+v)*(1-2*v))
μ = E/(2*(1+v))

# Integration

```

```

degree = 2*order
Ω = Triangulation(model)
dΩ = Measure(Ω, degree)
Γ_right = BoundaryTriangulation(model, tags=["RightBoundary"])
dΓ_right = Measure(Γ_right, degree)
t_vec = VectorValue(traction_val, 0.0)

# -- WEAK FORM (Direct implementation avoids 'evaluate!' errors) --
# a(u,v) = Integral of [ λ(div u)(div v) + 2μ(ε(u) : ε(v)) ]
a(u,v) = ∫( λ*(∇·v)*(∇·u) + 2*μ*(ε(v) ⊙ ε(u)) )dΩ
l(v) = ∫( v · t_vec )dΓ_right

# Solve
op = AffineFEOperator(a, l, U, V0)
uh = solve(op)

# Post-Process
# We define stress function purely for visualization
σ_func(ε) = λ*tr(ε)*one(ε) + 2*μ*ε
writevtk(Ω, "plate_results", cellfields=[{"u":>uh, "stress":>σ_func◦ε(uh)])
```

println("SUCCESS: Solution saved to 'plate\_results.vtu'")

```

Info      : Clearing all models and views...
Info      : Done clearing all models and views
Info      : Meshing 1D...
Info      : [ 0%] Meshing curve 1 (Line)
Info      : [ 20%] Meshing curve 2 (Line)
Info      : [ 30%] Meshing curve 3 (Line)
Info      : [ 40%] Meshing curve 4 (Line)
Info      : [ 60%] Meshing curve 5 (Circle)
Info      : [ 70%] Meshing curve 6 (Circle)
Info      : [ 80%] Meshing curve 7 (Circle)
Info      : [ 90%] Meshing curve 8 (Circle)
Info      : Done meshing 1D (Wall 0.004004s, CPU 0s)
Info      : Meshing 2D...
Info      : Meshing surface 1 (Plane, Frontal-Delaunay)
Info      : Done meshing 2D (Wall 0.0877483s, CPU 0.09375s)
Info      : 1242 nodes 2491 elements
Info      : Writing 'plate_hole_robust.msh'...
Info      : Done writing 'plate_hole_robust.msh'
Mesh saved to plate_hole_robust.msh
Info      : Reading 'plate_hole_robust.msh'...
Info      : 18 entities
Info      : 1241 nodes
Info      : 2358 elements
Info      : Done reading 'plate_hole_robust.msh'
SUCCESS: Solution saved to 'plate_results.vtu'
```

## QUESTION 5

# Finite Element Analysis of a 3D Cantilever Beam

## 1. Problem Definition

We analyze the stress distribution of a 3D cantilever beam subjected to a point load at its free end.

### Geometric and Material Parameters:

- **Dimensions:** Length  $L = 1000$  mm, Width  $W = 250$  mm, Depth  $H = 200$  mm.
- **Material:** Concrete.
- **Young's Modulus:**  $E = 25000$  N/mm<sup>2</sup> = 25 GPa.
- **Poisson's Ratio:**  $\nu = 0.2$ .
- **Boundary Condition:** Fixed support at  $x = 0$  (Left face).
- **Loading:** Point load  $P = -1000$  N in the vertical ( $z$ ) direction at the top center of the free end ( $x = 1000$ ).

## 2. Governing Equations (3D Linear Elasticity)

The problem is governed by the 3D equations of equilibrium, kinematic relations, and Hooke's Law.

### 1. Equilibrium Equation:

$$\nabla \cdot \boldsymbol{\sigma} + \mathbf{f} = \mathbf{0}$$

Where  $\mathbf{f}$  represents body forces (neglected here).

### 2. Strain-Displacement Relation:

$$\boldsymbol{\epsilon} = \frac{1}{2}(\nabla \mathbf{u} + (\nabla \mathbf{u})^T)$$

### 3. Constitutive Law (Isotropic Linear Elasticity):

$$\boldsymbol{\sigma} = \lambda(\text{tr } \boldsymbol{\epsilon})\mathbf{I} + 2\mu\boldsymbol{\epsilon}$$

Where Lamé parameters are defined as:

$$\lambda = \frac{E\nu}{(1+\nu)(1-2\nu)}, \quad \mu = \frac{E}{2(1+\nu)}$$

### 4. Boundary Conditions:

- **Dirichlet (Fixed):**  $\mathbf{u} = (0, 0, 0)$  at  $x = 0$ .
- **Neumann (Point Load):** A concentrated force is applied at a specific node on the face  $x = L$ . In weak form, this appears as a term in the linear functional:  $l(\mathbf{v}) = \mathbf{v}(x_{load}) \cdot \mathbf{P}$ .

## 3. Finite Element Analysis Steps in Abaqus (3D Cantilever)

## 1. Part Module

- **Create Part:** 3D, Deformable, Solid, Extrusion.
- **Sketch:** Draw a rectangle  $250 \times 200$  mm.
- **Extrude:** Depth = 1000 mm.

## 2. Property Module

- **Material:** Name **Concrete**.
  - **Elastic:**  $E = 25000$  MPa,  $\nu = 0.2$ .
- **Section:** Create Section, Solid, Homogeneous.
- **Assign Section:** Select the entire beam volume.

## 3. Assembly Module

- **Instance:** Create Instance (Dependent).

## 4. Step Module

- **Create Step:** Static, General.

## 5. Load Module

- **Boundary Condition (Fixed):**
  - Select the face at  $x = 0$ .
  - Set  $U1 = U2 = U3 = 0$  (ENCAS - Encastre).
- **Load (Point Load):**
  - Create **Concentrated Force**.
  - Select the midpoint node on the top edge of the face at  $x = 1000$ .
  - **Magnitude:**  $CF3 = -1000$  (Negative Z direction).
  - *Alternative:* Apply **Surface Traction** on the end face with magnitude  $1000/(250 \times 200) = 0.02$  MPa for smoother results.

## 6. Mesh Module

- **Element Type:** Standard, Linear, 3D Stress (**C3D8R** - 8-node linear brick, reduced integration).
- **Seeds:** Global Size  $\approx 20 - 50$  mm.
- **Mesh:** Mesh the part.

## 7. Job Module

- **Submit:** Create and submit the job.

## 8. Visualization

- **Deformation:** Plot  $\text{U}$  (Magnitude or U3). Theoretical max deflection  $\delta = \frac{PL^3}{3EI}$ .
- **Stress:** Plot  $S$ , Mises or  $S_{11}$  (Bending Stress).

```

using Gridap
using GridapGmsh
import GridapGmsh: gmsh

# =====
# 1. ROBUST GMSH INITIALIZATION
# =====
if GridapGmsh.gmsh.isInitialized() == 1
    GridapGmsh.gmsh.finalize()
end
GridapGmsh.gmsh.initialize()
gmsh.clear()
gmsh.model.add("Cantilever_Beam_3D")

# =====
# 2. GEOMETRY & MESH (3D Box)
# =====
L = 1000.0
W = 250.0 # Width (Y direction)
H = 200.0 # Height (Z direction)
mesh_size = 50.0 # Element size

# Create 3D Box using OpenCASCADE kernel
# addBox(x, y, z, dx, dy, dz)
box_tag = gmsh.model.occ.addBox(0, 0, 0, L, W, H)
gmsh.model.occ.synchronize()

# -- Physical Groups --
# We need to find the surfaces to apply boundary conditions.
# Surface at x=0 is Fixed.
# We will apply the point load at a specific node later, or distributed on a sm
# For simplicity in 3D FEM without nodal selection complexity, we often apply a
# distributed load on the small end face or handle the point load via Dirac del
# However, Gridap allows point loads if we identify the entity.

# Let's group the Fixed Face (x=0) and the Free Face (x=L)
surfaces = gmsh.model.getEntities(2)
fixed_face = Int32[]
free_face = Int32[]

for surf in surfaces
    dim, tag = surf
    # Get bounding box
    xmin, ymin, zmin, xmax, ymax, zmax = gmsh.model.getBoundingBox(dim, tag)

    # Check for x=0 (Fixed)
    if abs(xmin - 0.0) < 1e-3 && abs(xmax - 0.0) < 1e-3

```

```

        push!(fixed_face, tag)
    end

    # Check for x=L (Free)
    if abs(xmin - L) < 1e-3 && abs(xmax - L) < 1e-3
        push!(free_face, tag)
    end
end

volume_tags = [tag for (dim, tag) in gmsh.model.getEntities(3)]

gmsh.model.addPhysicalGroup(2, fixed_face, 1, "FixedWall")
gmsh.model.addPhysicalGroup(2, free_face, 2, "FreeEnd")
gmsh.model.addPhysicalGroup(3, volume_tags, 3, "BeamVolume")

# -- Mesh Generation --
gmsh.option.setNumber("Mesh.CharacteristicLengthMin", mesh_size)
gmsh.option.setNumber("Mesh.CharacteristicLengthMax", mesh_size)
gmsh.model.mesh.generate(3)

# Write to file
msh_file = "cantilever_3d.msh"
gmsh.write(msh_file)
GridapGmsh.gmsh.finalize()
println("3D Mesh saved to $msh_file")

# =====
# 3. FINITE ELEMENT ANALYSIS
# =====

model = GmshDiscreteModel(msh_file)

# Parameters
E = 25000.0 # N/mm^2 (MPa)
v = 0.2
P_val = -1000.0 # N (Downward force)

# Lamé parameters
λ = (E*v)/((1+v)*(1-2*v))
μ = E/(2*(1+v))

# FE Spaces
order = 1
# Note: VectorValue{3} for 3D analysis
reffe = ReferenceFE(lagrangian, VectorValue{3,Float64}, order)
V0 = TestFESpace(model, reffe; dirichlet_tags=["FixedWall"])
U = TrialFESpace(V0, VectorValue(0.0,0.0,0.0))

# Integration
degree = 2*order
Ω = Triangulation(model)

```

```

dΩ = Measure(Ω, degree)

# -- Handling Point Load in 3D --
# Applying a true point load in continuum elements can result in singularities.
# However, for this problem, we can distribute the 1000N over the 'FreeEnd' face
# as a traction to avoid singularity issues while preserving static equivalence
# Area of end face = 250 * 200 = 50000 mm^2
traction_z = P_val / (W * H)
traction_vec = VectorValue(0.0, 0.0, traction_z)

Γ_free = BoundaryTriangulation(model, tags=["FreeEnd"])
dΓ_free = Measure(Γ_free, degree)

# Weak Form
# 3D Elasticity: σ = λ tr(ε) I + 2μ ε
a(u,v) = ∫( λ*(∇·v)*(∇·u) + 2*μ*(ε(v) ⊙ ε(u)) )dΩ
l(v) = ∫( v · traction_vec )dΓ_free

# Solve
op = AffineFEOperator(a, l, U, V0)
uh = solve(op)

# Post-Process
σ_func(ε) = λ*tr(ε)*one(ε) + 2*μ*ε
writevtk(Ω, "cantilever_3d_solution", cellfields={"u":>uh, "stress":>σ_func*ε(u))

println("Analysis Complete! Visualization saved to 'cantilever_3d_solution.vtu'")

```

```

Info      : Clearing all models and views...
Info      : Done clearing all models and views
Info      : Meshing 1D...
Info      : [  0%] Meshing curve 1 (Line)
Info      : [ 10%] Meshing curve 2 (Line)
Info      : [ 20%] Meshing curve 3 (Line)
Info      : [ 30%] Meshing curve 4 (Line)
Info      : [ 40%] Meshing curve 5 (Line)
Info      : [ 50%] Meshing curve 6 (Line)
Info      : [ 60%] Meshing curve 7 (Line)
Info      : [ 60%] Meshing curve 8 (Line)
Info      : [ 70%] Meshing curve 9 (Line)
Info      : [ 80%] Meshing curve 10 (Line)
Info      : [ 90%] Meshing curve 11 (Line)
Info      : [100%] Meshing curve 12 (Line)
Info      : Done meshing 1D (Wall 0.00411296s, CPU 0.015625s)
Info      : Meshing 2D...
Info      : [  0%] Meshing surface 1 (Plane, Frontal-Delaunay)
Info      : [ 20%] Meshing surface 2 (Plane, Frontal-Delaunay)
Info      : [ 40%] Meshing surface 3 (Plane, Frontal-Delaunay)
Info      : [ 60%] Meshing surface 4 (Plane, Frontal-Delaunay)
Info      : [ 70%] Meshing surface 5 (Plane, Frontal-Delaunay)
Info      : [ 90%] Meshing surface 6 (Plane, Frontal-Delaunay)
Info      : Done meshing 2D (Wall 0.0602319s, CPU 0.046875s)
Info      : Meshing 3D...

```

```

Info : 3D Meshing 1 volume with 1 connected component
Info : Tetrahedrizing 512 nodes...
Info : Done tetrahedrizing 520 nodes (Wall 0.016202s, CPU 0.015625s)
Info : Reconstructing mesh...
Info : - Creating surface mesh
Info : - Identifying boundary edges
Info : - Recovering boundary
Info : Done reconstructing mesh (Wall 0.0367458s, CPU 0.03125s)
Info : Found volume 1
Info : It. 0 - 0 nodes created - worst tet radius 2.20134 (nodes removed 0)
Info : 3D refinement terminated (643 nodes total):
Info : - 0 Delaunay cavities modified for star shapeness
Info : - 0 nodes could not be inserted
Info : - 2233 tetrahedra created in 0.0199947 sec. (111679 tets/s)
Info : 0 node relocations
Info : Done meshing 3D (Wall 0.100944s, CPU 0.109375s)
Info : Optimizing mesh...
Info : Optimizing volume 1
Info : Optimization starts (volume = 5e+07) with worst = 0.00372219 / average = 0.000000
Info : 0.00 < quality < 0.10 : 11 elements
Info : 0.10 < quality < 0.20 : 24 elements
Info : 0.20 < quality < 0.30 : 17 elements
Info : 0.30 < quality < 0.40 : 39 elements
Info : 0.40 < quality < 0.50 : 46 elements
Info : 0.50 < quality < 0.60 : 109 elements
Info : 0.60 < quality < 0.70 : 376 elements
Info : 0.70 < quality < 0.80 : 578 elements
Info : 0.80 < quality < 0.90 : 659 elements
Info : 0.90 < quality < 1.00 : 369 elements
Info : 52 edge swaps, 1 node relocations (volume = 5e+07): worst = 0.306189
Info : No ill-shaped tets in the mesh :-)
Tetgen: 2023-07-14 10:10:10

```

## QUESTION 5

# Finite Element Analysis of a Cantilever Beam with Distributed Load

## 1. Problem Definition

We model the stress distribution of a 3D rectangular cantilever beam subjected to a uniform distributed pressure on its top surface.

### Geometric and Material Parameters:

- **Dimensions:** Length  $L = 1000$  mm, Width  $W = 250$  mm, Depth  $H = 200$  mm.
- **Material:** Concrete.
- **Young's Modulus:**  $E = 25000$  N/mm<sup>2</sup>.

- **Poisson's Ratio:**  $\nu = 0.2$ .
- **Boundary Condition:** Fixed support at  $x = 0$  (Left face).
- **Loading:** Uniform distributed load (pressure)  $q = 1000 \text{ N/mm}^2$  applied downwards on the top surface ( $z = H$ ).
  - *Note: 1000 N/mm<sup>2</sup> is an extremely high pressure (1 GPa), but we follow the problem statement exactly.*

## 2. Governing Equations (3D Linear Elasticity)

### 1. Equilibrium Equation:

$$\nabla \cdot \boldsymbol{\sigma} + \mathbf{f} = \mathbf{0}$$

### 2. Strain-Displacement:

$$\boldsymbol{\epsilon} = \frac{1}{2}(\nabla \mathbf{u} + (\nabla \mathbf{u})^T)$$

### 3. Constitutive Law:

$$\boldsymbol{\sigma} = \lambda(\text{tr } \boldsymbol{\epsilon})\mathbf{I} + 2\mu\boldsymbol{\epsilon}$$

### 4. Boundary Conditions:

- **Dirichlet:**  $\mathbf{u} = \mathbf{0}$  on  $\Gamma_{fixed}$  ( $x = 0$ ).
- **Neumann:**  $\boldsymbol{\sigma} \cdot \mathbf{n} = \mathbf{t}$  on  $\Gamma_{top}$  ( $z = H$ ), where  $\mathbf{t} = (0, 0, -1000)$ .
- **Free:**  $\boldsymbol{\sigma} \cdot \mathbf{n} = \mathbf{0}$  on all other surfaces.

## 3. Finite Element Analysis Steps in Abaqus (UDL Case)

### 1. Part Module

- **Create Part:** 3D, Deformable, Solid, Extrusion.
- **Sketch:** Rectangle  $250 \times 200$ . Extrude depth 1000.

### 2. Property Module

- **Material:** Elastic, Isotropic.  $E = 25000$ ,  $\nu = 0.2$ .
- **Section:** Solid, Homogeneous. Assign to volume.

### 3. Assembly Module

- **Instance:** Create Dependent Instance.

### 4. Step Module

- **Create Step:** Static, General.

### 5. Load Module

- **Boundary Condition (Fixed):**

- Select face  $x = 0$ .
- Encastre ( $U_1 = U_2 = U_3 = UR_1 = UR_2 = UR_3 = 0$ ).
- **Load (Pressure):**
  - Create Load -> **Pressure**.
  - Select the **Top Surface** ( $z = 200$ ).
  - **Magnitude:** 1000 (Note: Abaqus pressure is positive inwards/compressive. If you want it downward like gravity, input positive 1000 on the top face). **6.**

### Mesh Module

- **Element:** C3D8R (Hexahedral).
- **Seed:** Size approx 40-50.
- **Mesh Part.**

## 7. Job Module

- **Submit Job.**

## 8. Visualization

- Check **U3** (Vertical deflection).
- Check **S11** (Bending stress). Max stress should be at the fixed support top/bottom fibers.

```

using Gridap
using GridapGmsh
import GridapGmsh: gmsh

# =====
# 1. ROBUST GMSH INITIALIZATION
# =====
if GridapGmsh.gmsh.isInitialized() == 1
    GridapGmsh.gmsh.finalize()
end
GridapGmsh.gmsh.initialize()
gmsh.clear()
gmsh.model.add("Cantilever_UDL")

# =====
# 2. GEOMETRY & MESH
# =====
L = 1000.0
W = 250.0 # Width (Y)
H = 200.0 # Height (Z)
mesh_size = 40.0

# Create Box
gmsh.model.occ.addBox(0, 0, 0, L, W, H)
gmsh.model.occ.synchronize()

```

```

# -- Physical Groups --
# We need:
# 1. Fixed Wall (x=0)
# 2. Top Surface (z=H) -> For the UDL

surfaces = gmsh.model.getEntities(2)
fixed_face = Int32[]
top_face   = Int32[]

for surf in surfaces
    dim, tag = surf
    xmin, ymin, zmin, xmax, ymax, zmax = gmsh.model.getBoundingBox(dim, tag)

    # Check for Fixed Wall (x ~ 0)
    if abs(xmin - 0.0) < 1e-3 && abs(xmax - 0.0) < 1e-3
        push!(fixed_face, tag)
    end

    # Check for Top Surface (z ~ H)
    if abs(zmin - H) < 1e-3 && abs(zmax - H) < 1e-3
        push!(top_face, tag)
    end
end

volume_tags = [tag for (dim, tag) in gmsh.model.getEntities(3)]

gmsh.model.addPhysicalGroup(2, fixed_face, 1, "FixedWall")
gmsh.model.addPhysicalGroup(2, top_face, 2, "TopSurface")
gmsh.model.addPhysicalGroup(3, volume_tags, 3, "BeamVolume")

# -- Mesh Generation --
gmsh.option.setNumber("Mesh.CharacteristicLengthMin", mesh_size)
gmsh.option.setNumber("Mesh.CharacteristicLengthMax", mesh_size)
gmsh.model.mesh.generate(3)

msh_file = "cantilever_udl.msh"
gmsh.write(msh_file)
GridapGmsh.gmsh.finalize()
println("Mesh saved to $msh_file")

# =====
# 3. FINITE ELEMENT ANALYSIS
# =====

model = GmshDiscreteModel(msh_file)

# Parameters
E = 25000.0
v = 0.2
pressure_val = -1000.0 # N/mm^2 (Downward pressure)

```

```

# Lamé parameters
λ = (E*v)/((1+v)*(1-2*v))
μ = E/(2*(1+v))

# FE Spaces
order = 1
reffe = ReferenceFE(lagrangian, VectorValue{3,Float64}, order)
V0 = TestFESpace(model, reffe; dirichlet_tags=["FixedWall"])
U = TrialFESpace(V0, VectorValue(0.0,0.0,0.0))

# Integration
degree = 2*order
Ω = Triangulation(model)
dΩ = Measure(Ω, degree)

# -- LOAD DEFINITION --
# We apply the load on the "TopSurface" tag
Γ_top = BoundaryTriangulation(model, tags=["TopSurface"])
dΓ_top = Measure(Γ_top, degree)
traction_vec = VectorValue(0.0, 0.0, pressure_val)

# Weak Form
a(u,v) = ∫( λ*(∇·v)*(∇·u) + 2*μ*(ε(v) ⊙ ε(u)) )dΩ
l(v) = ∫( v · traction_vec )dΓ_top

# Solve
op = AffineFEOperator(a, l, U, V0)
uh = solve(op)

# Post-Process
σ_func(ε) = λ*tr(ε)*one(ε) + 2*μ*ε
writevtk(Ω, "cantilever_udl_solution", cellfields=[ "u"=>uh, "stress"=>σ_func◦ε(

println("Analysis Complete! Visualization saved to 'cantilever_udl_solution.vtu"

```

```

Info      : Clearing all models and views...
Info      : Done clearing all models and views
Info      : Meshing 1D...
Info      : [  0%] Meshing curve 1 (Line)
Info      : [ 10%] Meshing curve 2 (Line)
Info      : [ 20%] Meshing curve 3 (Line)
Info      : [ 30%] Meshing curve 4 (Line)
Info      : [ 40%] Meshing curve 5 (Line)
Info      : [ 50%] Meshing curve 6 (Line)
Info      : [ 60%] Meshing curve 7 (Line)
Info      : [ 60%] Meshing curve 8 (Line)
Info      : [ 70%] Meshing curve 9 (Line)
Info      : [ 80%] Meshing curve 10 (Line)
Info      : [ 90%] Meshing curve 11 (Line)
Info      : [100%] Meshing curve 12 (Line)
Info      : Done meshing 1D (Wall 0.00209188s, CPU 0.015625s)
Info      : Meshing 2D...

```

```
Info : [  0%] Meshing surface 1 (Plane, Frontal-Delaunay)
Info : [ 20%] Meshing surface 2 (Plane, Frontal-Delaunay)
Info : [ 40%] Meshing surface 3 (Plane, Frontal-Delaunay)
Info : [ 60%] Meshing surface 4 (Plane, Frontal-Delaunay)
Info : [ 70%] Meshing surface 5 (Plane, Frontal-Delaunay)
Info : [ 90%] Meshing surface 6 (Plane, Frontal-Delaunay)
Info : Done meshing 2D (Wall 0.0651231s, CPU 0.0625s)
Info : Meshing 3D...
Info : 3D Meshing 1 volume with 1 connected component
Info : Tetrahedrizing 790 nodes...
Info : Done tetrahedrizing 798 nodes (Wall 0.011375s, CPU 0.015625s)
Info : Reconstructing mesh...
Info : - Creating surface mesh
Info : - Identifying boundary edges
Info : - Recovering boundary
Info : Done reconstructing mesh (Wall 0.0229959s, CPU 0.015625s)
Info : Found volume 1
Info : It. 0 - 0 nodes created - worst tet radius 2.74761 (nodes removed 0)
Info : 3D refinement terminated (1104 nodes total):
Info : - 0 Delaunay cavities modified for star shapeness
Info : - 0 nodes could not be inserted
Info : - 4227 tetrahedra created in 0.030165 sec. (140129 tets/s)
Info : 0 node relocations
Info : Done meshing 3D (Wall 0.107453s, CPU 0.109375s)
Info : Optimizing mesh...
Info : Optimizing volume 1
Info : Optimization starts (volume = 5e+07) with worst = 0.0281426 / average = 0.000100000
Info : 0.00 < quality < 0.10 : 13 elements
Info : 0.10 < quality < 0.20 : 34 elements
Info : 0.20 < quality < 0.30 : 37 elements
Info : 0.30 < quality < 0.40 : 64 elements
```