1.

A. Strong - formulations:

Stress - Strain relation (Hook's Law):

$$\sigma_{xx} = \frac{E}{1-\nu^2}(\varepsilon_{xx} + \nu\varepsilon_{yy}), \quad \sigma_{yy} = \frac{E}{1-\nu^2}(\varepsilon_{yy} + \nu\varepsilon_{xx})$$

$$\sigma_{xy} = G\gamma_{xy}.$$

among. $G = \frac{E}{2(1+\nu)}$, $\varepsilon_{xx} = \frac{\partial u}{\partial x}$. $\varepsilon_{yy} = \frac{\partial v}{\partial y}$, $\gamma_{xy} = \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}$.

Boundary Conditions: $\sigma_{xx} = 10\ kPa$. $\sigma_{xy} = 0$.

when $y=0$, $v=0$, $\sigma_{xy} = 0$.

when $x=0$, $u=0$, $\sigma_{xy} = 0$.

radio $= 0.5m$. $\sigma_{rr} = 0$, $\sigma_{\theta r} = 0$.

B. Weak Form:

$$\int_\Omega \delta u \left(\frac{\partial \sigma_{xx}}{\partial x} + \frac{\partial \sigma_{xy}}{\partial y}\right) d\Omega = 0, \quad \int_\Omega \delta v \left(\frac{\partial \sigma_{xy}}{\partial x} + \frac{\partial \sigma_{yy}}{\partial y}\right) d\Omega = 0.$$

$$\Rightarrow \int_\Omega \sigma_{xx}\frac{\partial \delta u}{\partial x} + \sigma_{xy}\frac{\partial \delta u}{\partial y}\, d\Omega - \int_{\Gamma_t} \delta u\, t_x\, d\Gamma = 0.$$

$$\int_\Omega \sigma_{xy}\frac{\partial \delta v}{\partial x} + \sigma_{yy}\frac{\partial \delta v}{\partial y}\, d\Omega - \int_{\Gamma_t} \delta v\, t_y\, d\Gamma = 0.$$

$\Gamma_t$ is boundary.

C. Galerkin formulation

$$u(x,y) \approx \sum_{i=1}^{N} u_i \psi_i(x,y) \quad v(x,y) \approx \sum_{i=1}^{N} v_i \psi_i(x,y)$$

Weak form $\Rightarrow \int_\Omega C\,\varepsilon(u_h)\cdot\varepsilon(\delta u)\, d\Omega = \int_{\Gamma_t} \delta u \cdot t\, d\Gamma$

among that: $$C = \frac{E}{1-\nu^2}\begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \frac{1-\nu}{2} \end{bmatrix}$$

BCs: Dirichlet BC: $u=0$, at $x=0$, $v=0$ at $y=0$.

Neumann Conditions: $\sigma_{xx} = 10kPa$ at $x=0$. and $\sigma_{xy}=0$.

$\sigma_{rr}=0$, $\sigma_{r\theta}=0$ at hole boundary.

2. The implementation of the element Stiffness matrix.

For the FEM for 2D elasticity problem, I use 3-node linear triangular elements.

a. Numerical Integration:

The stiffness matrix is computed using Gaussian quadrature.
The code uses 3 Gauss points. And the Jacobian matrix ($J$) is computed to map the natural ~~coo~~ coordinates $(\xi, \eta)$ to global coordinates $(x, y)$.

b. Strain-Displacement Matrix (B).

$$B = \begin{bmatrix} \frac{\partial N_1}{\partial x} & 0 & \frac{\partial N_2}{\partial x} & 0 & \frac{\partial N_3}{\partial x} & 0 \\ 0 & \frac{\partial N_1}{\partial y} & 0 & \frac{\partial N_2}{\partial y} & 0 & \frac{\partial N_3}{\partial y} \\ \frac{\partial N_1}{\partial y} & \frac{\partial N_1}{\partial x} & \frac{\partial N_2}{\partial y} & \frac{\partial N_2}{\partial x} & \frac{\partial N_3}{\partial y} & \frac{\partial N_3}{\partial x} \end{bmatrix}.$$

c. Material Properties Matrix (D).

$$D = \frac{E}{1-\nu^2} \begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & \frac{1-\nu}{2} \end{bmatrix} \qquad \sigma = D\varepsilon.$$

d. Element Stiffness Matrix ($K_{loc}$).

$$K_{loc} = \int_{Element} B^T D B \det J \, dA.$$

e. Assembly into Global Stiffness Matrix.
The degrees of freedom (DOFs) for each node in ~~to~~ the element are mapped into the global system using the connectivity matrix IEN.

3. 带孔平板弹性力学问题的解析解

根据 M.H. Sadd, Elasticity: Theory, Applications, and Numerics, pp. 176-177 的介绍，我们得到了在极坐标下在恒定远场平面内具有无应力圆孔的无限平板的张力形式：

$$\sigma_{rr}(r,\theta)=\frac{T_x}{2}\left(1-\frac{R^2}{r^2}\right)+\frac{T_x}{2}\left(1-4\frac{R^2}{r^2}+3\frac{R^4}{r^4}\right)cos2\theta,$$

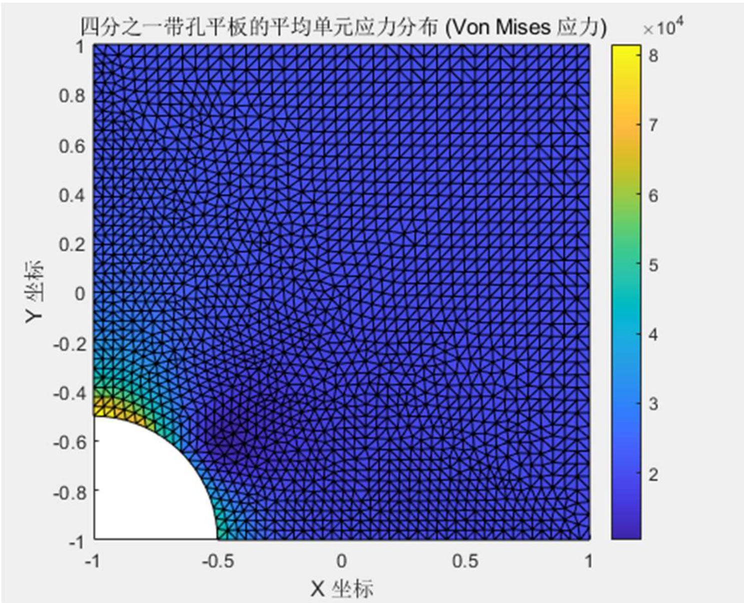
$$\sigma_{\theta\theta}(r,\theta)=\frac{T_x}{2}\left(1+\frac{R^2}{r^2}\right)-\frac{T_x}{2}\left(1+3\frac{R^4}{r^4}\right)cos2\theta,$$

$$\sigma_{r\theta}(r,\theta)=-\frac{T_x}{2}\left(1+2\frac{R^2}{r^2}-3\frac{R^4}{r^4}\right)sin2\theta.$$

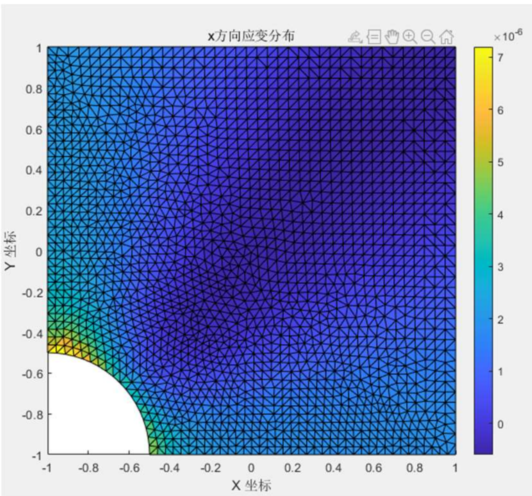
以上公式是在极坐标下的，我们需要用代码运行出其在笛卡尔坐标下的可视化效果。

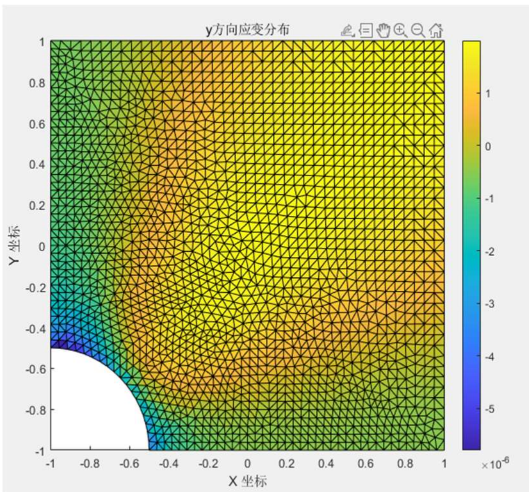
请运行我的代码：mesh.m 和 analytical.m

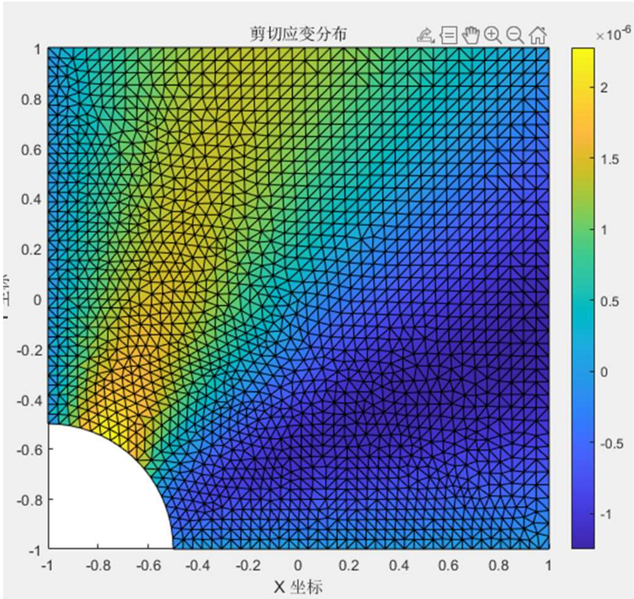
在代码中我针对应力的解析形式，获得了 x、y 方向应力，剪切应力，并计算 Von Mises 应力形式来表示单元中的平均应力。因为需要与 FEM 解进行对照，所以我是用了 Gmsh 获得的网格坐标。下图是展示效果：




同理，我根据弹性力学的本构关系，获得了应变图像。

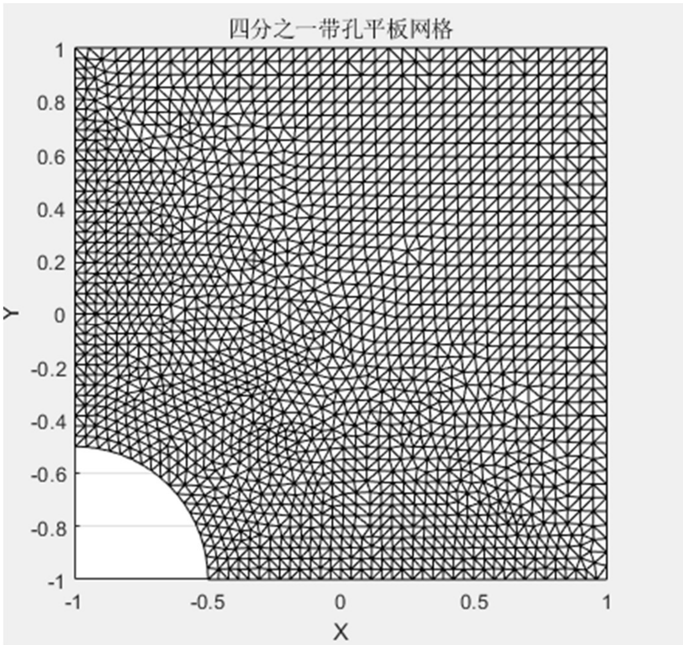


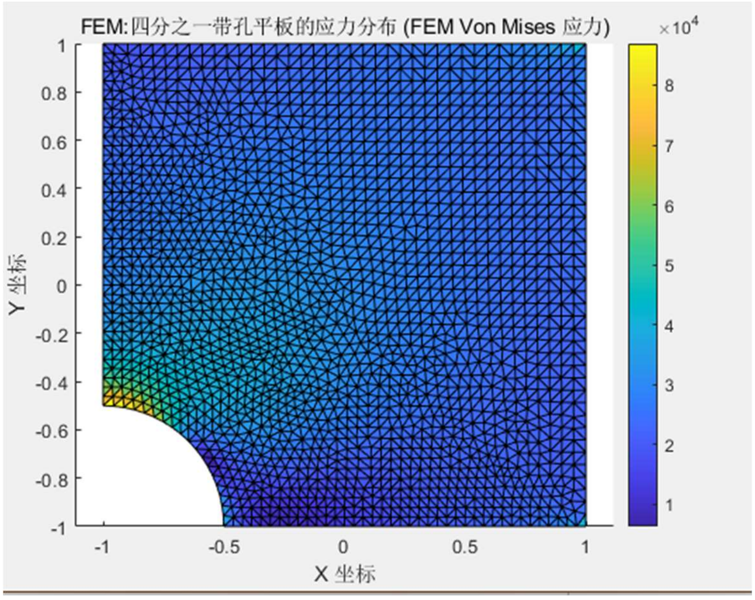


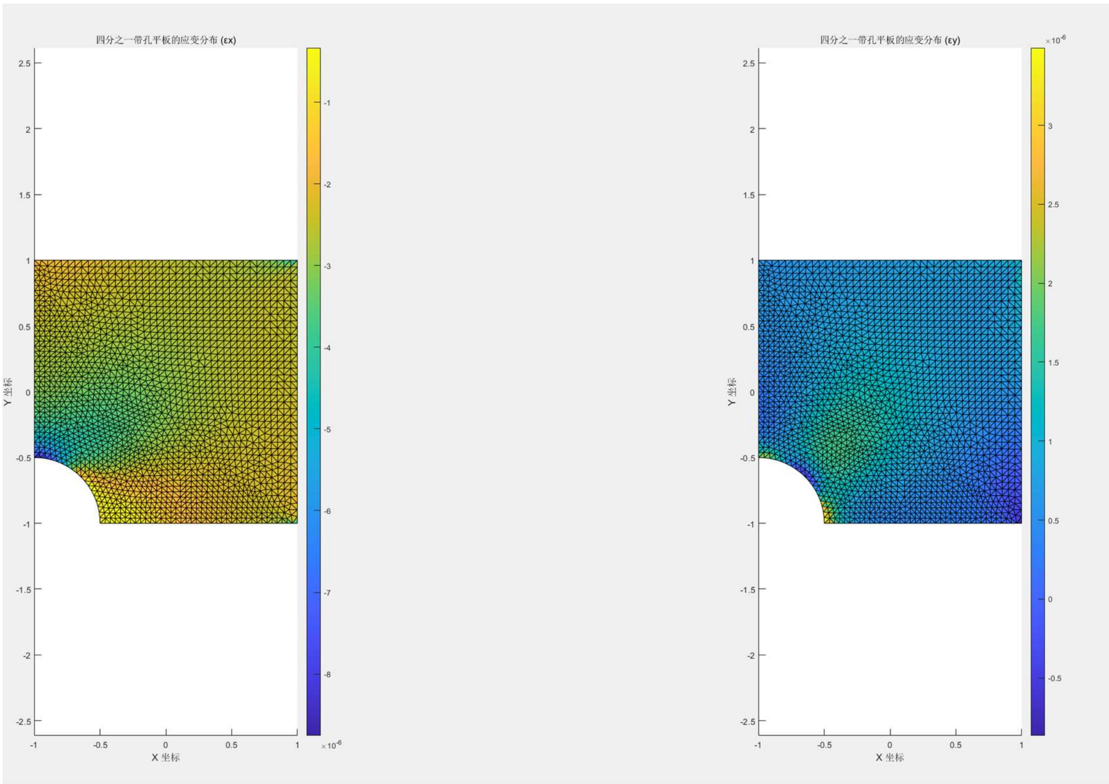

4. 有限元分析

我们先使用 Gmsh 画网格，并导入 matlab 中获得节点坐标。如图，我使用的是三角形网格，并合理处理各边界的网格尺寸。




然后运行我的代码：mesh.m 和 driver_proj_stress.m 和 driver_proj_strain.m 可以得到 FEM 方法的应力和应变分布云图。




经过直观对比，数值解和解析解的数量级和分布形式保持一直。需要进一步结合误差进行验证可靠性。