

MATLAB and Engineering Application

PDE Toolbox

Poisson's Equation on Unit Disk

$$c\left(x, t, u, \frac{\partial u}{\partial x}\right) \frac{\partial u}{\partial t} = x^{-m} \frac{\partial}{\partial x} \left(x^m f\left(x, t, u, \frac{\partial u}{\partial x}\right) \right) + s\left(x, t, u, \frac{\partial u}{\partial x}\right)$$

$m=0$ for Cartesian, 1 for cylindrical, 2 for spherical

$$u(x, t_0) = u_0(x)$$

Initial Conditions

$$p(x, t, u) + q(x, t) f\left(x, t, u, \frac{\partial u}{\partial x}\right) = 0$$


Boundary Conditions – one at each boundary

Poisson's Equation

$$\frac{\partial u}{\partial t} - \left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right) = 1$$

Boundary condition: $T=0$ @ boundary

Set up governing equation

 PDE Specification

Equation: $d*u'-div(c*grad(u))+a*u=f$

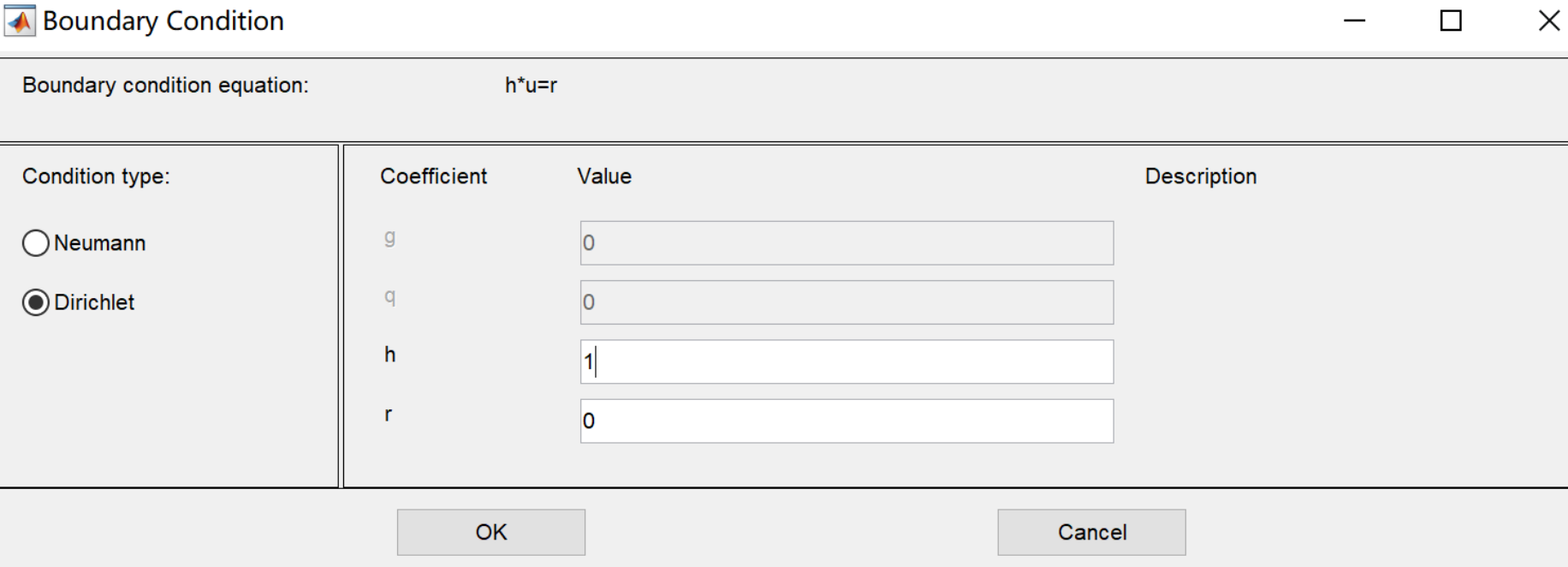
Type of PDE:	Coefficient	Value
<input type="radio"/> Elliptic	c	1.0
<input checked="" type="radio"/> Parabolic	a	0.0
<input type="radio"/> Hyperbolic	f	1
<input type="radio"/> Eigenmodes	d	1.0

OK

Cancel

Poisson's Equation

Set up governing equation



A software dialog box titled "Boundary Condition" with standard window controls (minimize, maximize, close). It contains a text field for the boundary condition equation, a section for selecting the condition type (Neumann or Dirichlet), and a table for defining coefficients and their values. The "Dirichlet" option is selected. The table has columns for "Coefficient", "Value", and "Description". The coefficients listed are g, q, h, and r, with values 0, 0, 1, and 0 respectively. The "h" value field is currently active with the cursor at the end of the input "1".

Boundary condition equation: $h*u=r$

Condition type:

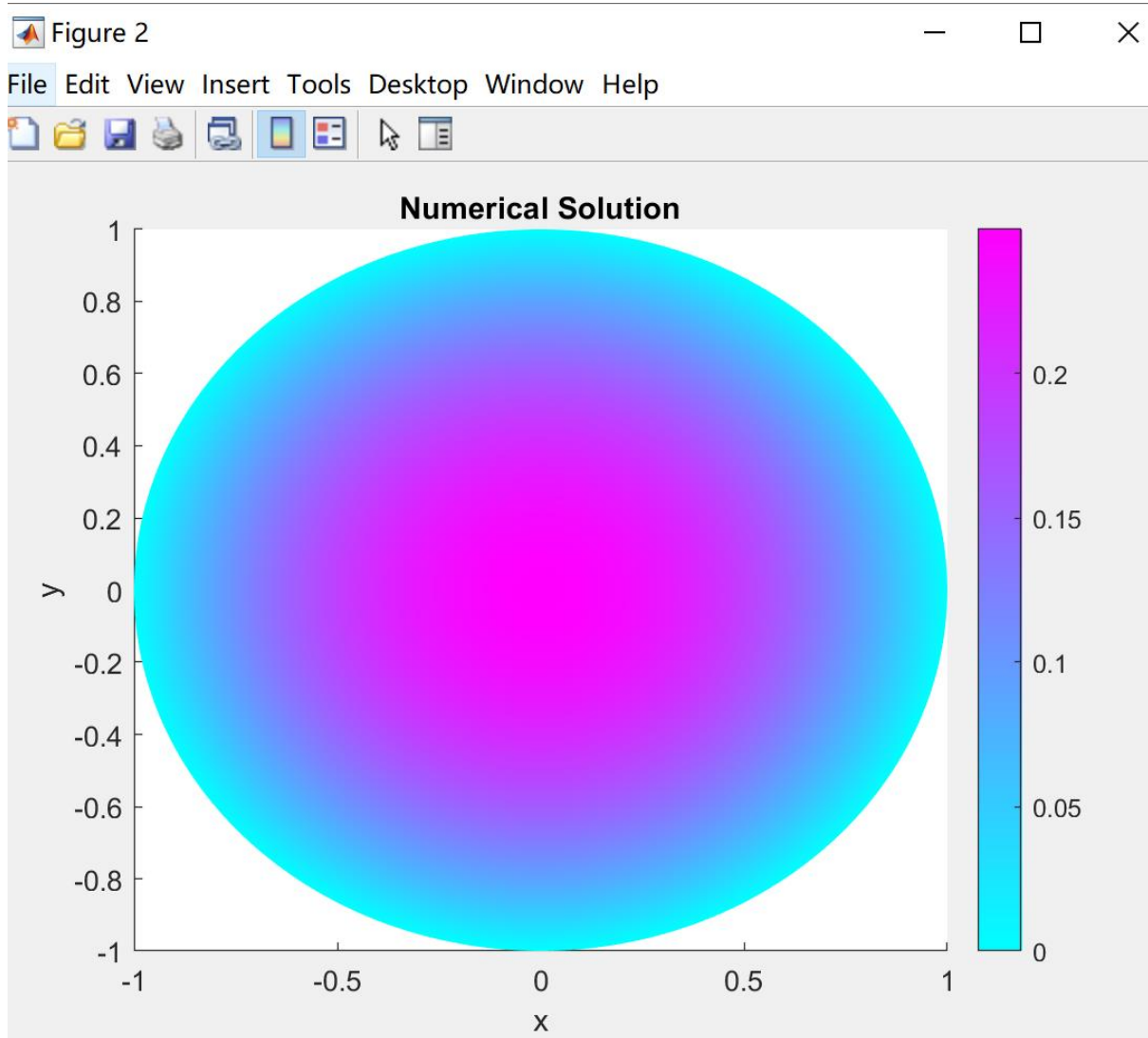
☐ Neumann

☒ Dirichlet

Coefficient	Value	Description
g	0	
q	0	
h	1	
r	0	

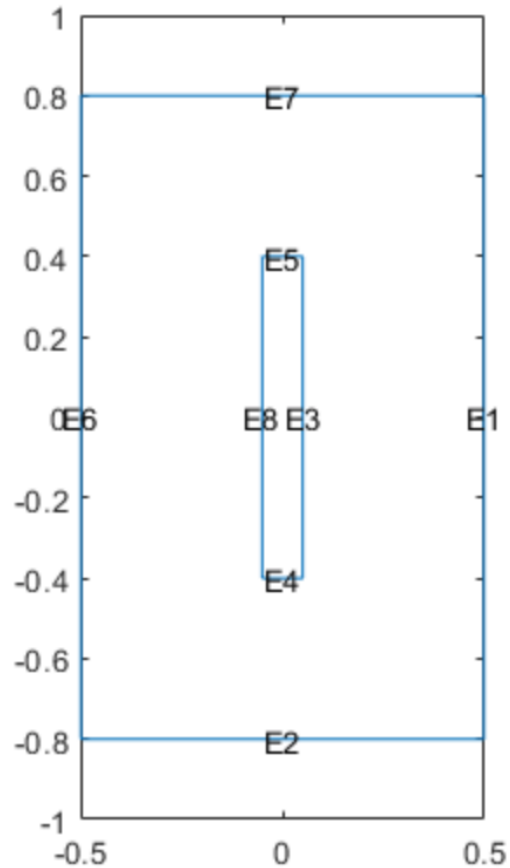
OK Cancel

Poisson's Equation



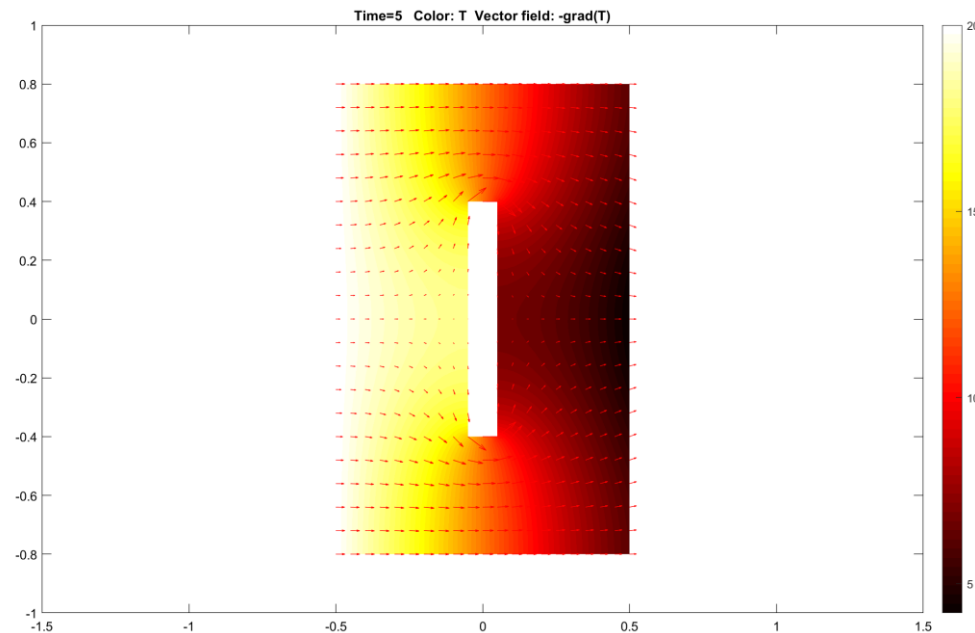
Heat Transfer in Block with Cavity

Consider a block containing a rectangular crack or cavity. The left side of the block is heated to 20 degrees centigrade. At the right side of the block, heat flows from the block to the surrounding air at a constant rate, for example $-10W/m^2$. All the other boundaries are insulated. The temperature in the block at the starting time $t_0=0$ is 0 degrees. The goal is to model the heat distribution during the first five seconds.

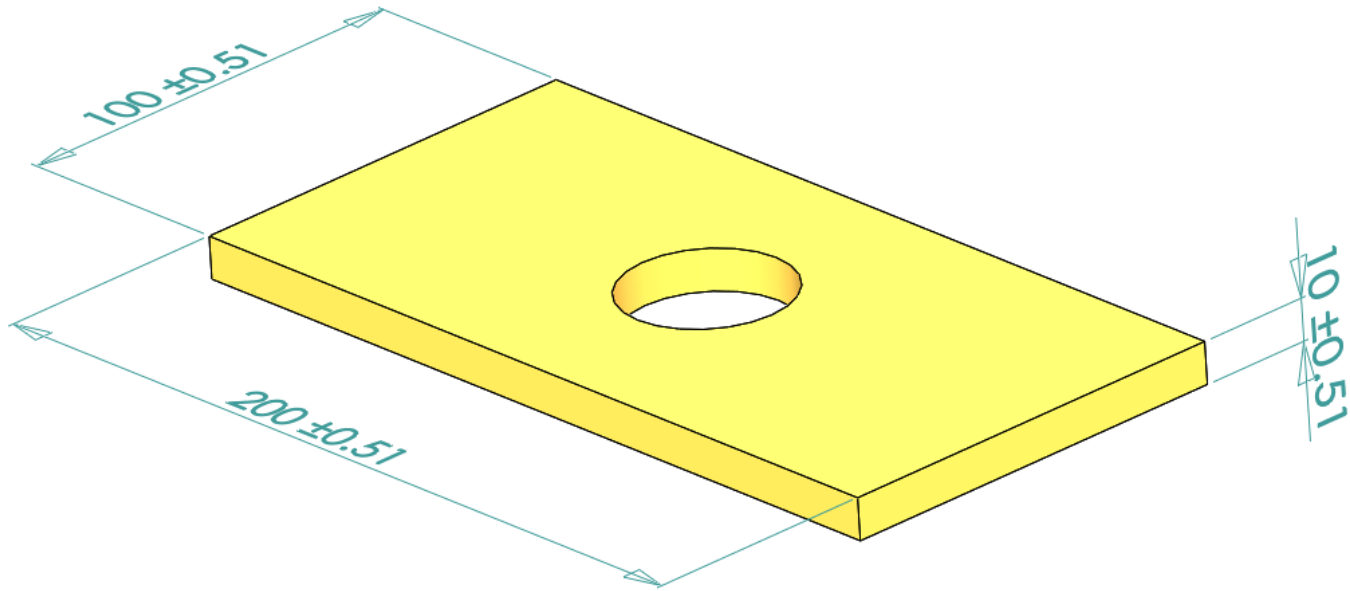


$$\rho c_p \frac{\partial T}{\partial t} - k \left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} \right) = 0$$

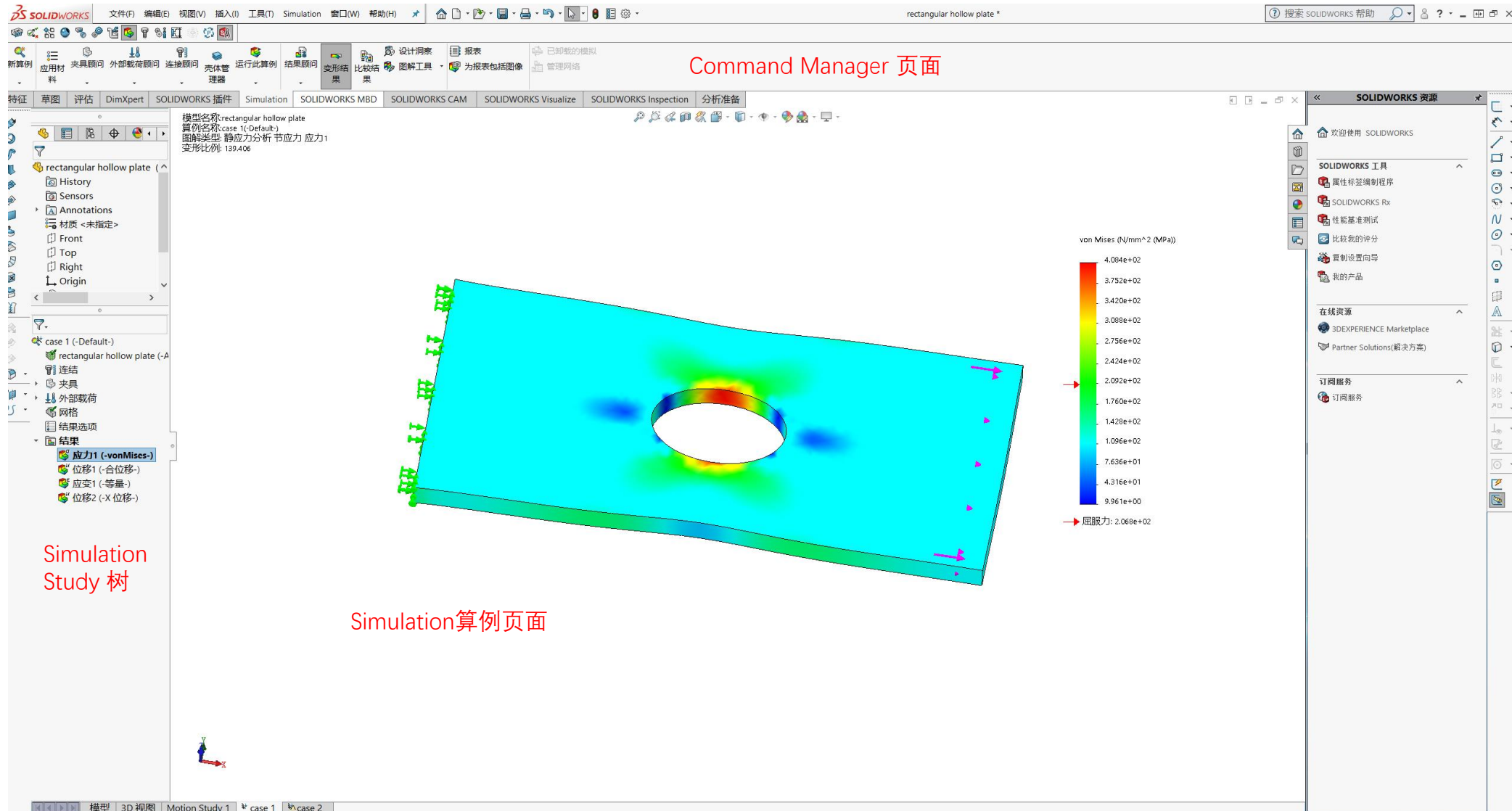
$$\rho = 1, c_p = 1, k = 1$$



Structural Mechanics



SolidWorks Simulation



Simulation flow

- New study (static analysis)
- Define material (AISI 304)
- Add fixture (set up boundary conditions)
- Apply external load

Fixture

Fixtures

The Fixture PropertyManager allows you to prescribe zero or non-zero displacements on vertices, edges, or faces for use with static, frequency, buckling, dynamic and nonlinear studies.

CONTENTS

[Applying a Restraint](#)

[Summary of Displacement Restraints](#)

[Preventing Rigid Body Motion](#)

[Fixed](#)

[Immovable](#)

[Using Reference Geometry](#)

You can use a selected reference geometry to apply restraints. The reference can be a plane, axis, edge, or face. Using this option you can prescribe restraints on faces and beam joints.

[On Flat Face](#)

[On Cylindrical Face](#)

[On Spherical Faces](#)

[Applying a Prescribed Displacement](#)

[Symmetry](#)

You can use symmetry to model a portion of the model instead of the full model. The results on the un-modeled portions are deducted from the modeled portions. When appropriate, taking advantage of symmetry can help you reduce the size of the problem and obtain more accurate results.

[Bearing Fixture PropertyManager](#)

The PropertyManager allows you to simulate the interaction between a shaft and the ground through a bearing. Since the components supporting the shaft are assumed to be much more rigid than the shaft, they can be considered as the ground. The feature is available for static, frequency, linear dynamic, and buckling studies.

PARENT TOPIC

[Loads and Restraints](#)

RELATED CONCEPTS

[Directional Loads and Restraints](#)

[Multiple Application of Displacement Restraints](#)

[Structural Loads](#)

[Connectors](#)

[Thermal Loads and Restraints](#)

[Function Curves](#)

RELATED TASKS

[Splitting a Face \(Split Tab\)](#)

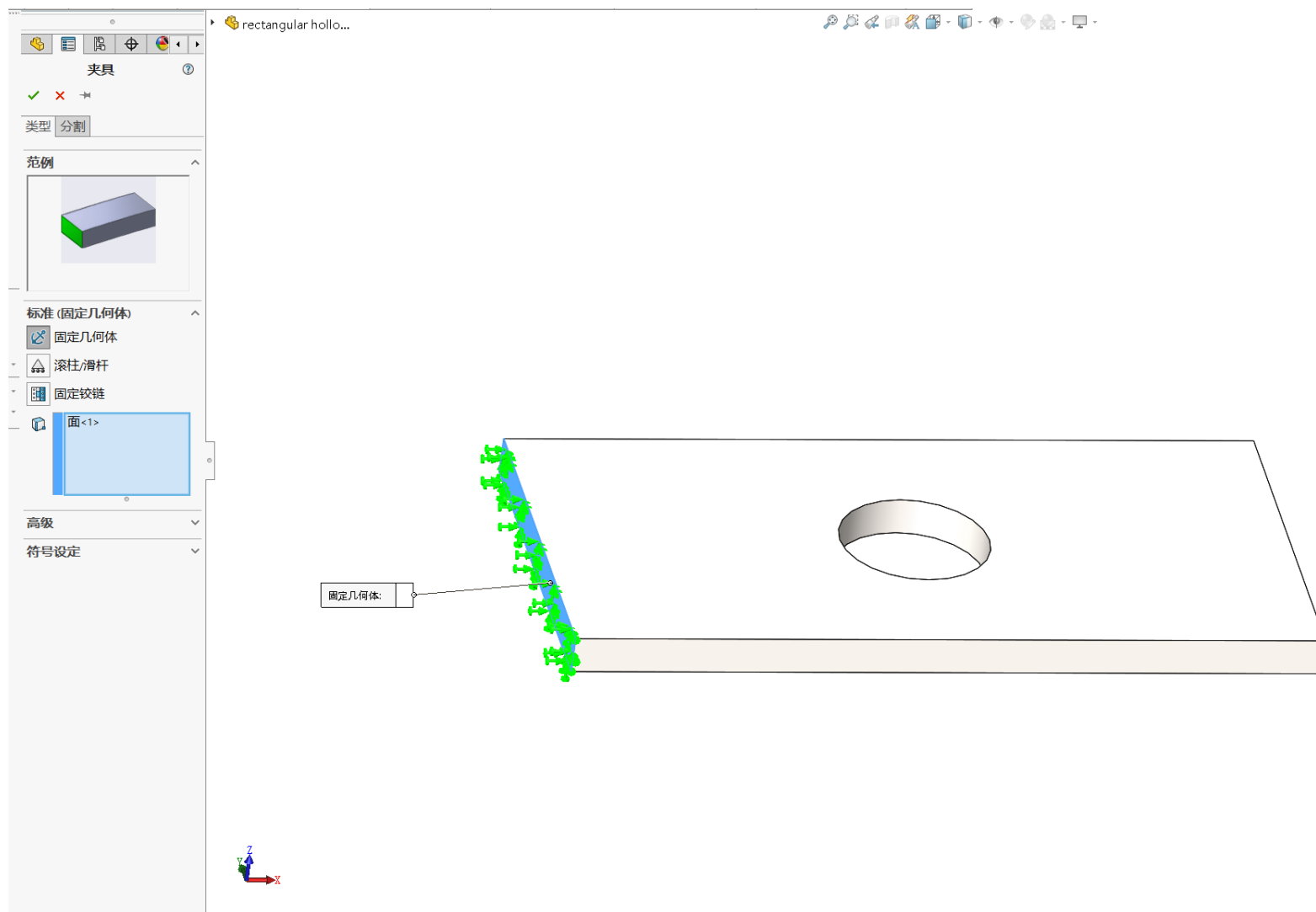
[Importing In-mold Stresses from SOLIDWORKS Plastics](#)

RELATED REFERENCE




[Split](#)

[Load Case Manager](#)

Immovable fixture



Apply external load



Structural Loads

Some type of loading, or a prescribed displacement, is required for static and buckling studies. Loads are optional for frequency studies.

CONTENTS

- Pressure**

You can apply uniform or nonuniform (variable) pressure to faces for use in structural (static, frequency, buckling, nonlinear and dynamic) studies.
- Force**

You can apply forces, moments, or torques with uniform distribution to faces, edges, reference points, vertices and beams in any direction for use in structural studies.
- Gravity**

The Gravity PropertyManager applies linear accelerations to a part or assembly document for use in structural and nonlinear analyses.
- Centrifugal**

Use the Centrifugal PropertyManager to apply angular velocity and acceleration to a part or an assembly in static, frequency, buckling, or nonlinear studies.
- Remote Loads and Restraints**

Remote loads, restraints, and masses can simplify a model.
- Bearing Loads**

Bearing loads develop between contacting cylindrical faces or edges of shells.
- Shrink Fitting**

Shrink fitting is encountered in many engineering designs. It refers to fitting an object into a slightly smaller cavity.
- Summary of Remote Load, Bearing Load, and Shrink Fit Load Options**
- Applying Loads and Restraints to Shells**
- Multiple Application of Structural Loads**
- Distributed Mass PropertyManager**

PARENT

- [Loads and Restraints](#)

RELATE

- [Directional Restraints](#)
- [Multiple Displacement Restraints](#)
- [Fixture Restraints](#)
- [Connections](#)
- [Thermal Restraints](#)
- [Functional Restraints](#)

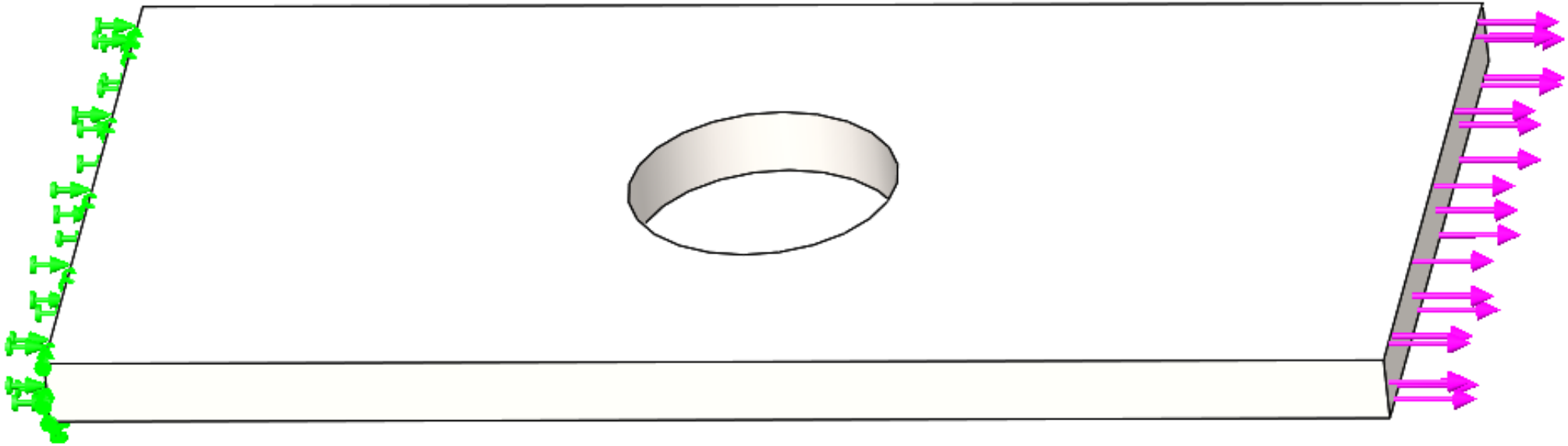
RELATE

- [Splitting Tab](#)
- [Import Stresses from SOLIDWORKS](#)

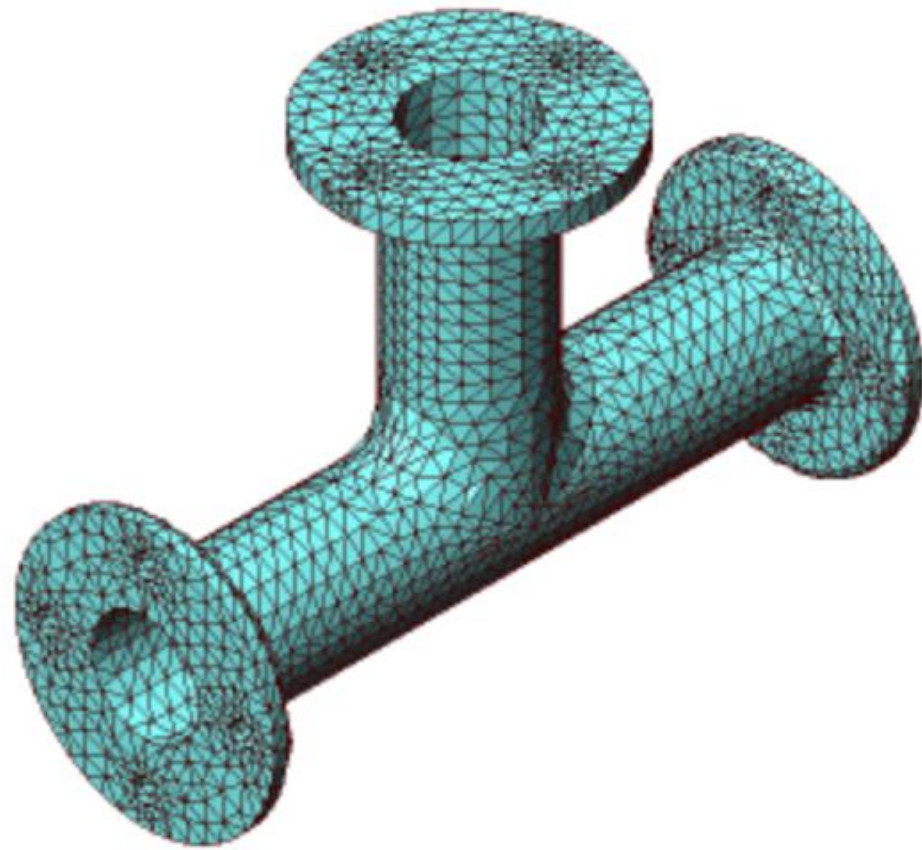
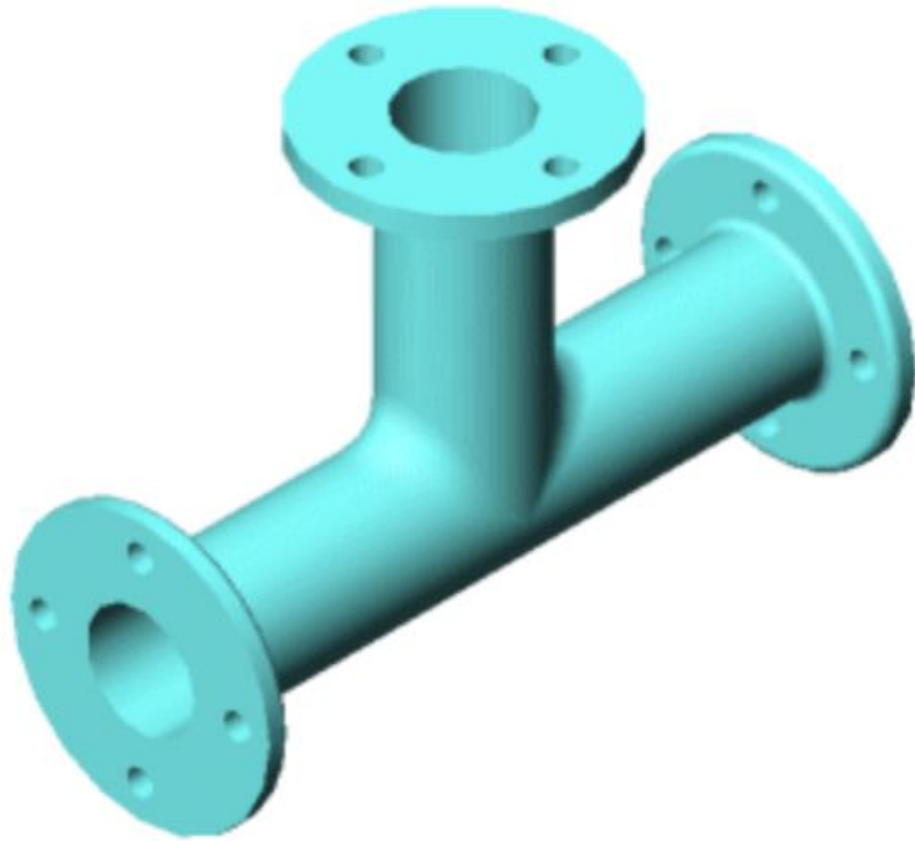
RELATE

- [Split](#)
- [Load Case](#)

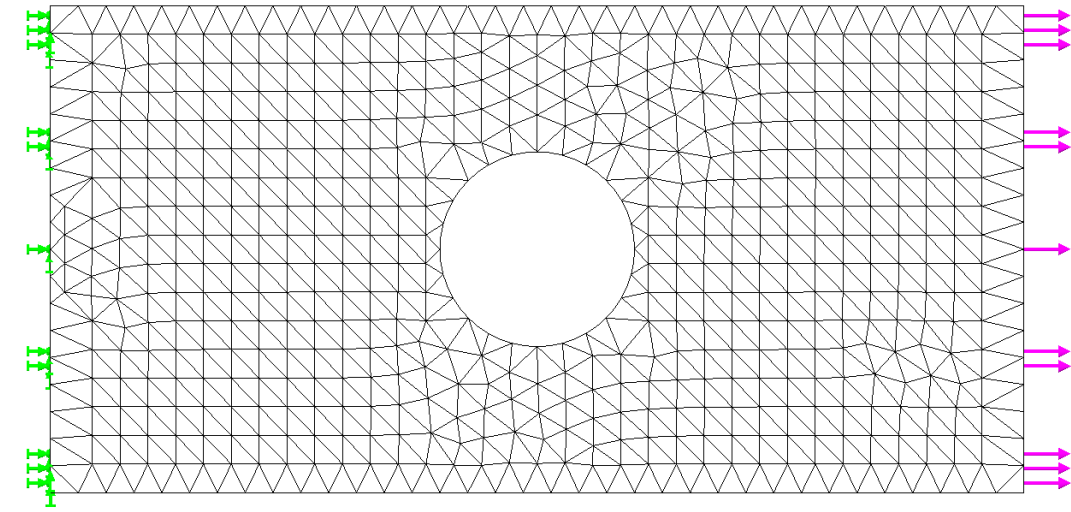
External force



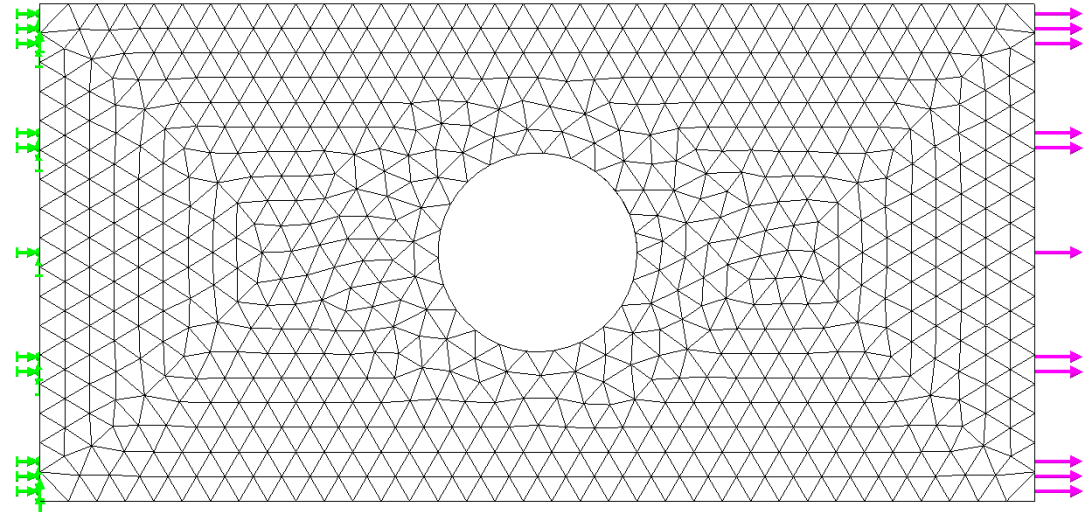
Meshing



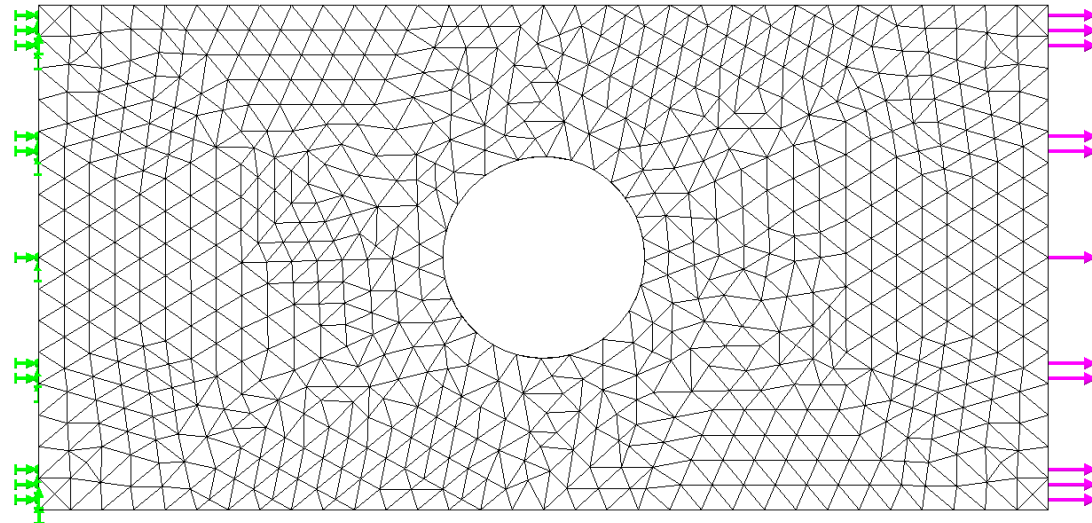
Meshing



Standard meshing

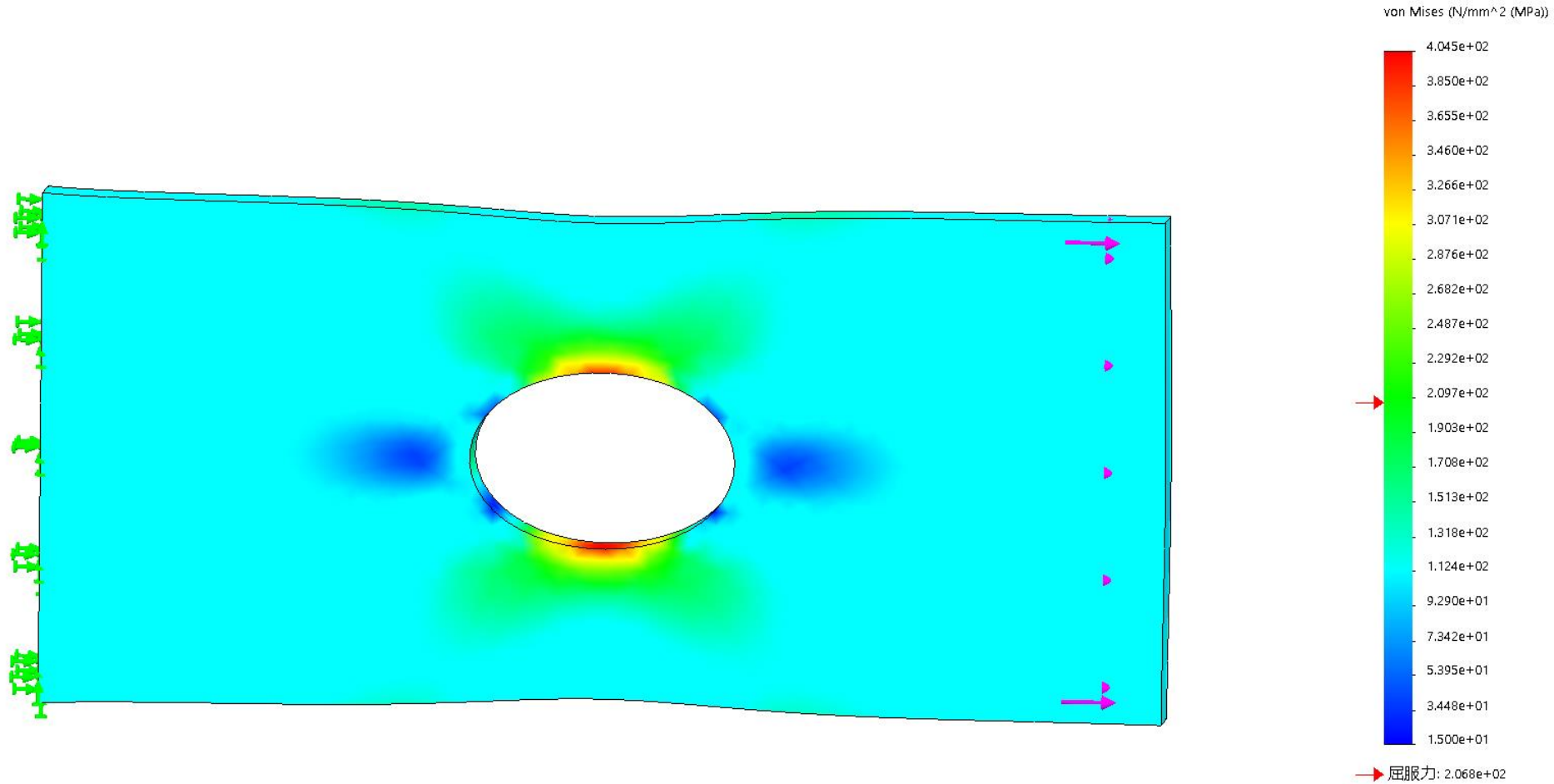


Blended Curvature Based Mesh



Curvature based mesh

Run and postprocessing

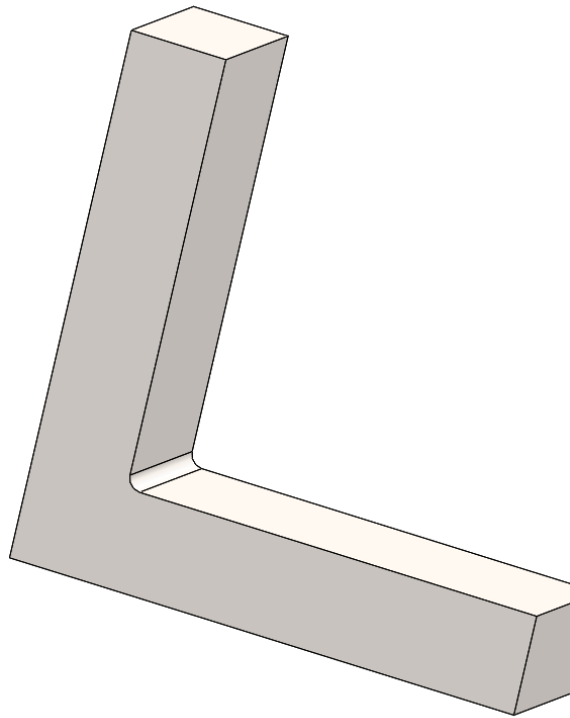


Comparison of results with various mesh system

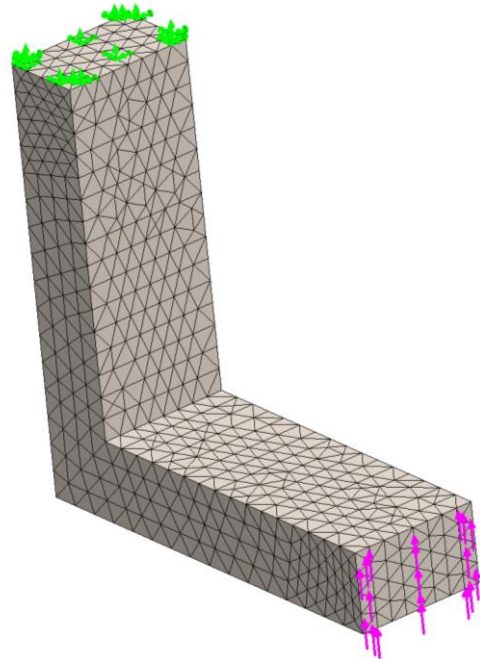
Mesh density	Maximum displacement/ mm	Maximum von Mises stress / MPa	Number of freedom	Number of elements	Number of nodes
Coarse analysis	0.143	387.54	6261	994	2144
Default analysis	0.1435	404.69	42822	8317	14455
Fine analysis	0.1435	415.35	298320	65728	100081

Nonuniform grid

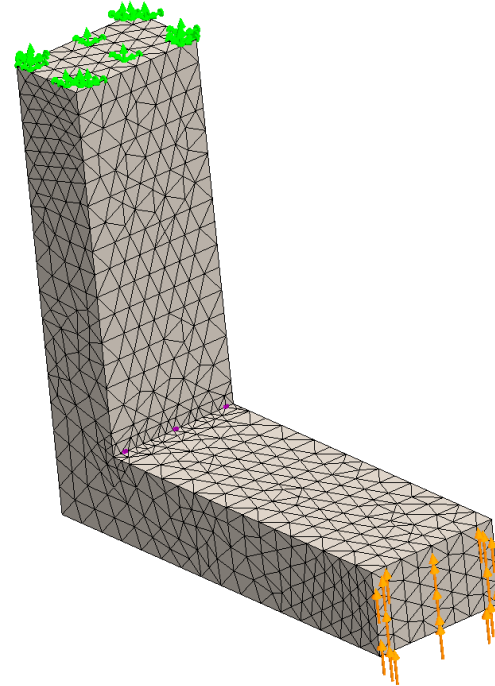
- In real-world problems, grids are rarely perfectly uniform. It would be very unwise to uniformly reduce the mesh size in a large model just because of local stress concentrations. Creating a large number of elements in a homogeneous area or an area where the stress changes gently will result in an increase in computational complexity, and the final result may not be much different. Different strategies can be used to control the mesh, such as tending to use a small mesh where the stress changes rapidly, and a larger mesh in areas with small changes.



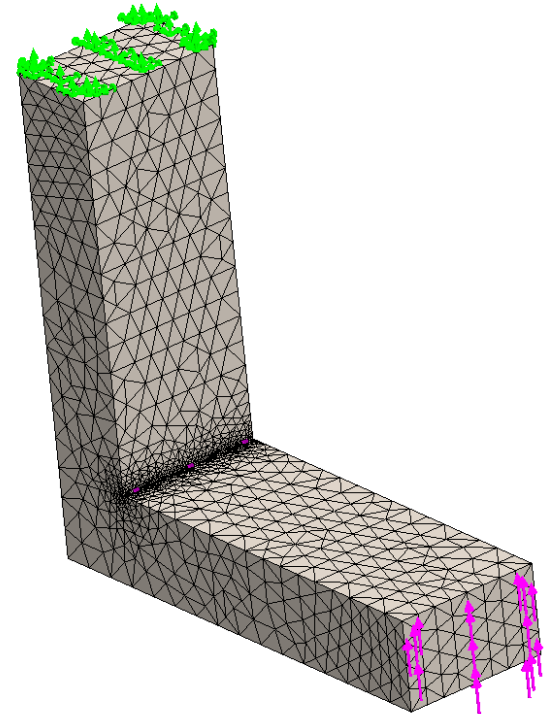
L-bracket



Mesh 1



Mesh 2



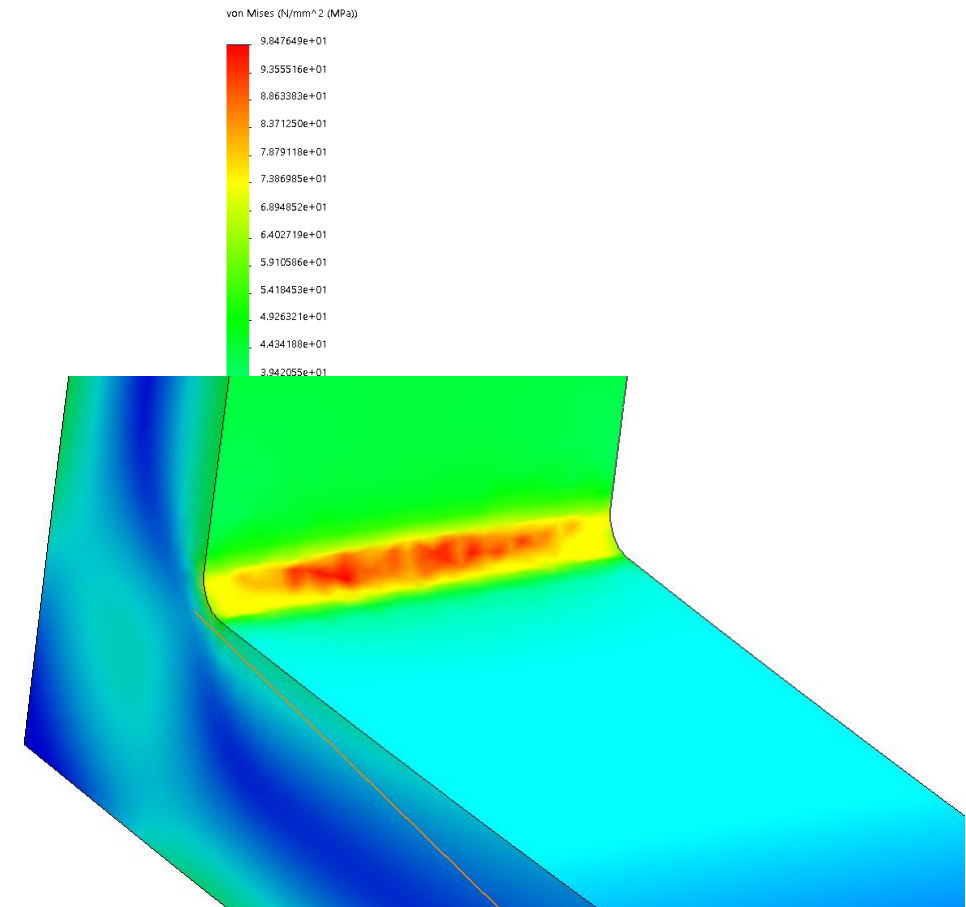
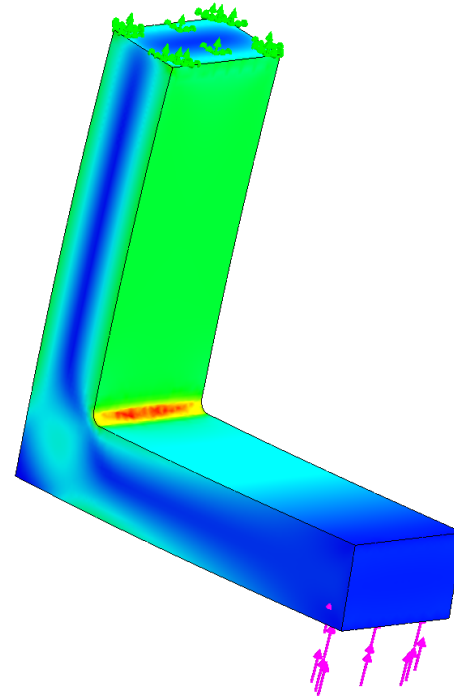
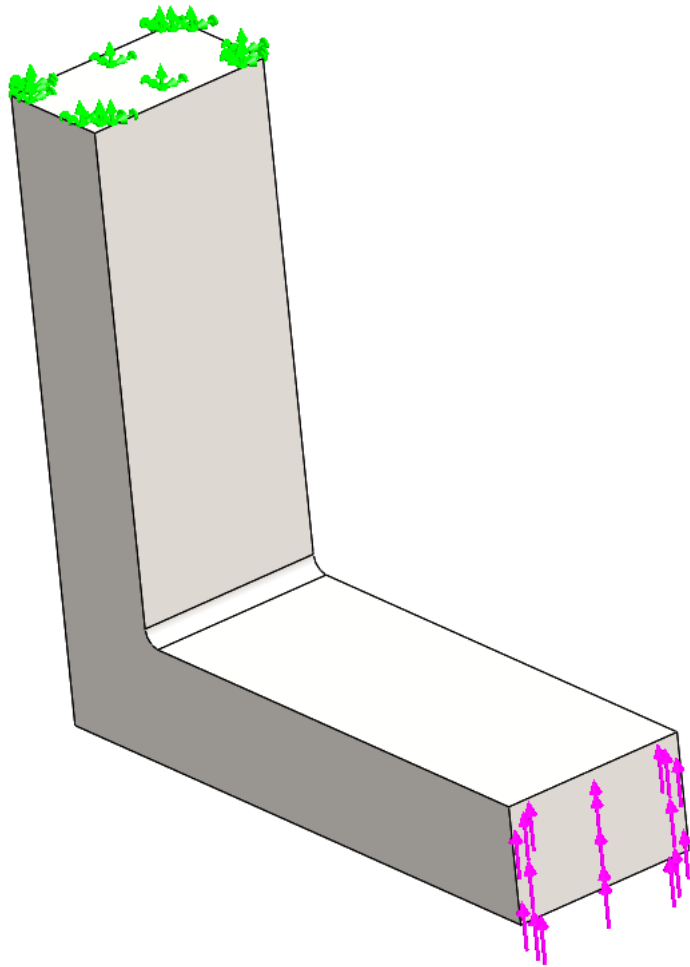
Mesh 3

Comparison of results

Example	Maximum displacement/mm	Maximum von Mises stress/MPa
Mesh 1	0.2874	60.664
Mesh 2	0.28835	117.858
Mesh 3	0.28856	166.62

- Each refinement of the mesh results in an increase in the maximum displacement and maximum stress values. The increase in displacement is minimal, and becomes smaller and smaller as the number of studies increases. Therefore, the displacement results can be considered to be convergent.
- As the mesh is progressively refined, the resulting stress values become larger and larger. Unlike displacement, which converges to a finite value, stress values diverge.
- The reason for the divergent stress results is not an error in the finite element model itself, but rather the finite element model is based on an incorrect mathematical model.
- According to the theory of elasticity, the stress at the sharp corners is infinite. The finite element model does not produce infinite stress results due to discretization errors that mask modeling errors.
- If the purpose is to determine the maximum stress value, it is a serious mistake to ignore the presence of rounded corners so that the model contains a sharp corner. If you want to understand the stress situation near a fillet, then it should be included in the model, no matter how small the fillet is.

Bracket analysis with rounded corners



Apply grid control around rounded corner

