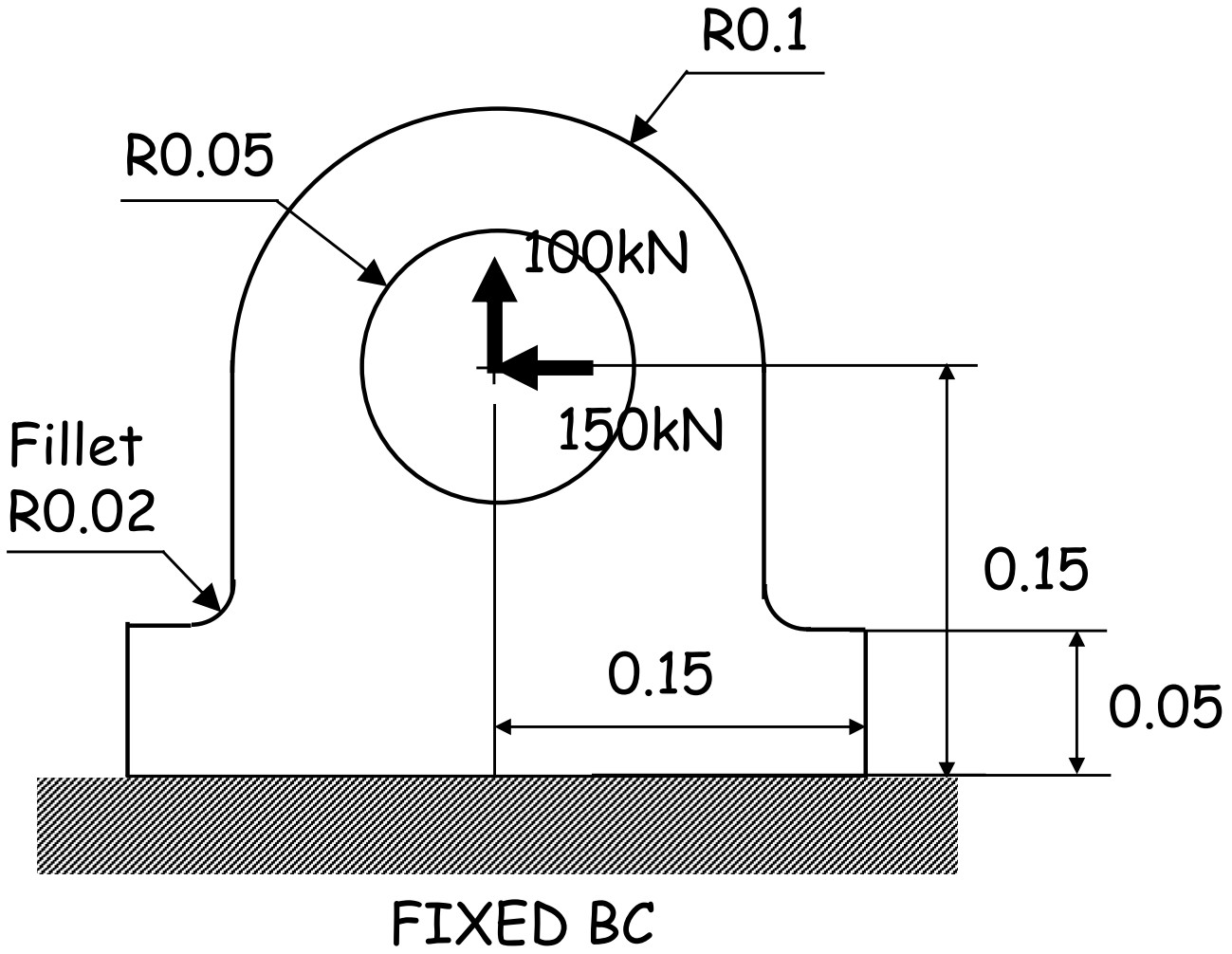
Tutorial 5-1:

Part Sketch / Geometric Constraints

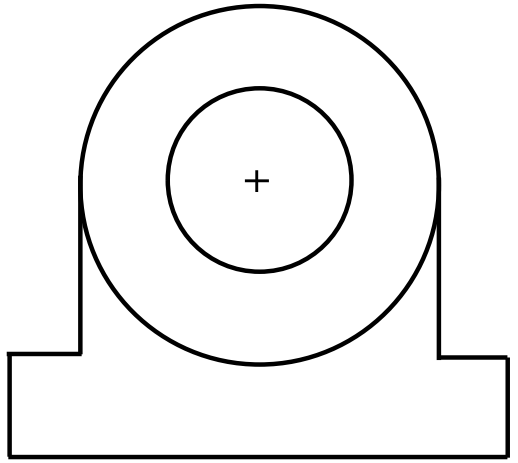
A BRACKET ANALYSIS

- A bracket with a shaft hole
  - E=210 Gpa, Poison ratio 0.3
  - Thickness t=0.1 m

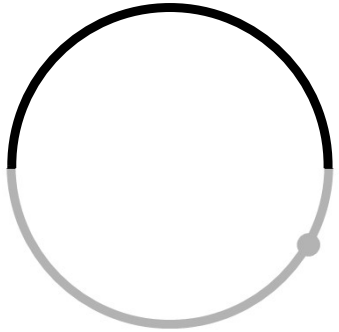


PART MODULE (SKETCH)

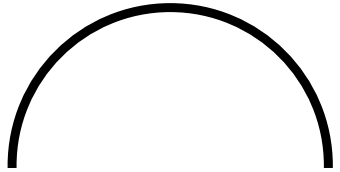
- Sketch
  - Draw outlines of the bottom of the bracket
- Tip
  - Starting and ending point of a circle is recognized as a dividing point



- Case 1

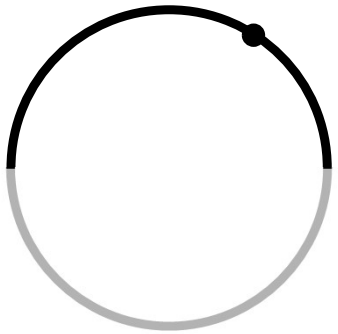


Delete the bottom half

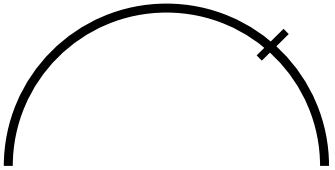


One geometric object

- Case 2



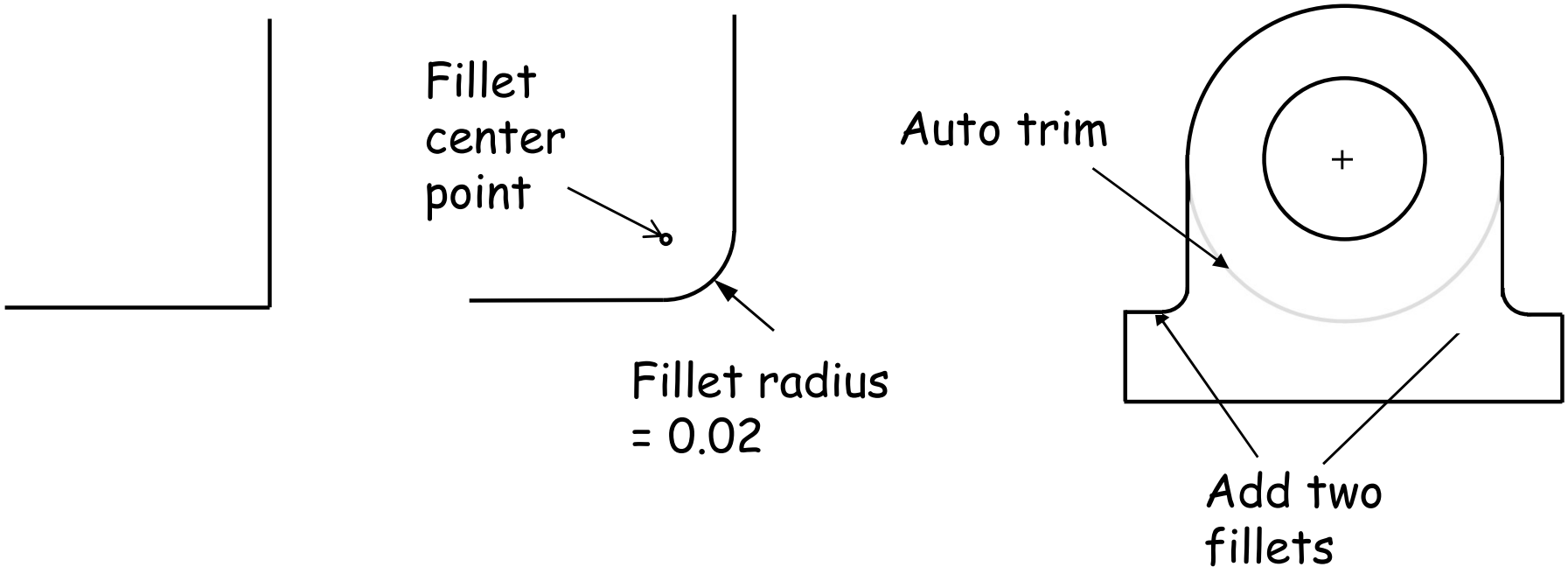
Delete the bottom half



Two geometric objects

PART MODULE (SKETCH)

- Sketch
  - Menu/Edit/Auto-Trim, delete half of the outer circle
  - Menu/Add/Fillet, add two fillets, radius of those fillets is 0.02

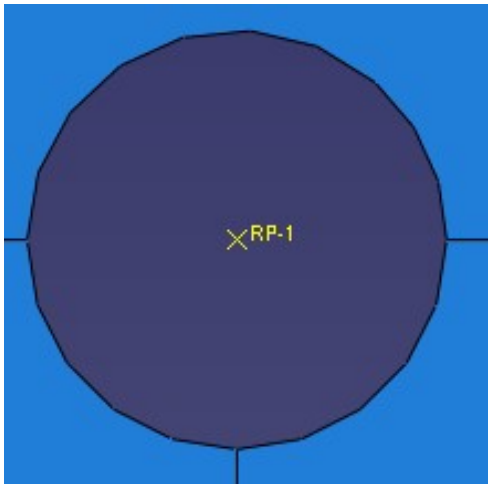


PROPERTY / ASSEMBLY / STEPS MODULES

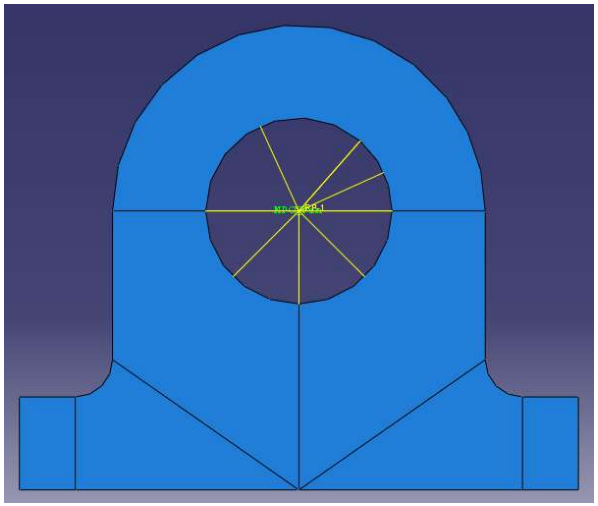
- Materials
  - Mechanical, Elasticity, Elastic
  - Young's modulus = 210E9, Poisson's ratio = 0.3
- Sections
  - Solid, Homogeneous
  - Set plane stress/strain thickness to 0.1 m
- Assign the section to the part
- Assembly, Instance
- Steps
  - Linear perturbation, Static

INTERACTION MODULE (MPC)

- How to apply loads at the center of shaft hole?
  - In Interaction module, Side tool bar/Create a reference point (RP) at the center of the shaft hole
  - Menu/Constraint/Create/MPC Constraint MPC (Multiple point constraints)
  - Select the RP as the MPC control point (master node)
  - Select the circumference of the hole shaft as the slave nodes
  - MPC type select as of Link



Reference Point



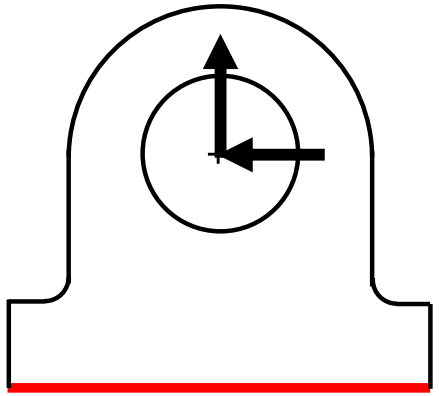
Applied Beam type MPCs

MPC Types

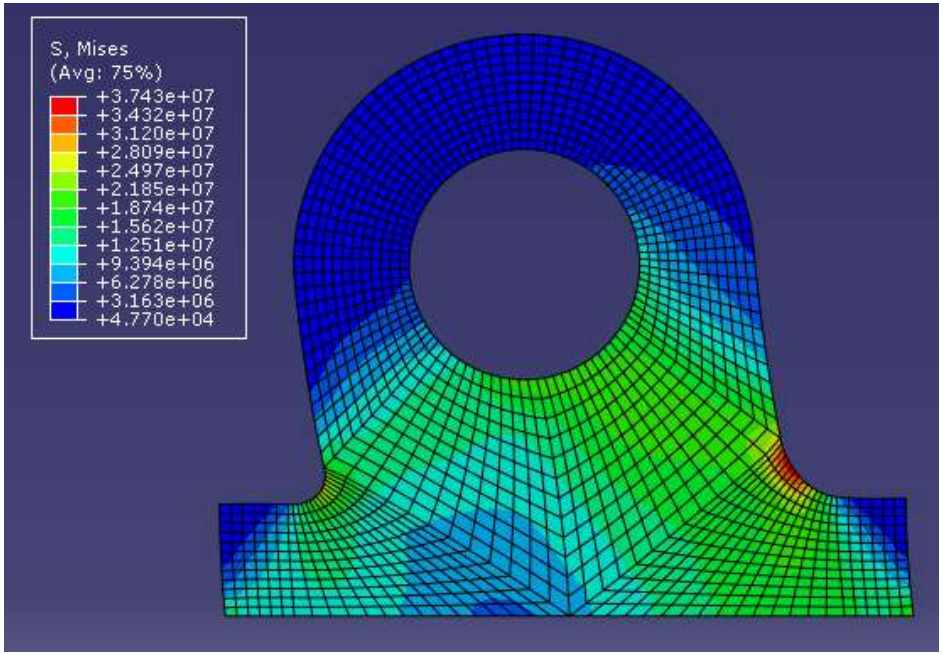
- Beam type
  - Provide a rigid beam between the master node and slave nodes
  - Constraint the "displacement" and "rotation" of the master node to the "displacement" and "rotation" of the slave nodes.
  - Distant between the master and slave nodes remain the same
- Pin type
  - Constraint equal global displacements between the master node and slave nodes
  - Constraint the "displacement" of the master node to the "displacement" of the slave nodes.
- Link type
  - a pinned rigid link between each slave node and the control point

LOADS / JOB / VISUALIZATION MODULES

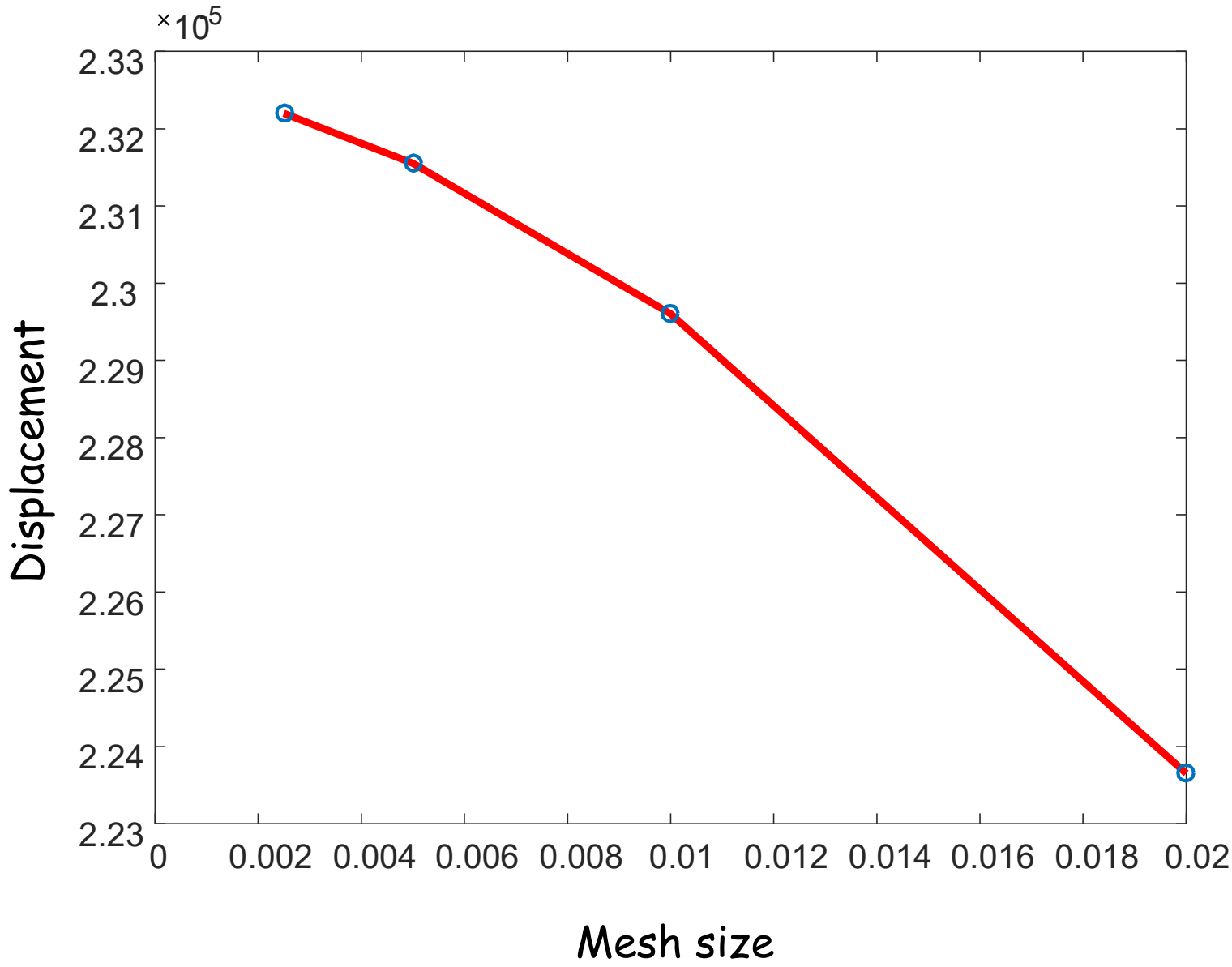
- Loads
  - Mechanical, Concentrated force, Uniform, CF1 = -150kN CF2 = 100kN
- BCs
  - Initial, ENCASTRE
- Analysis, Create Job, Data Check, Submit
- Results
  - Max Von Mises 37 MPa



FIXED BC

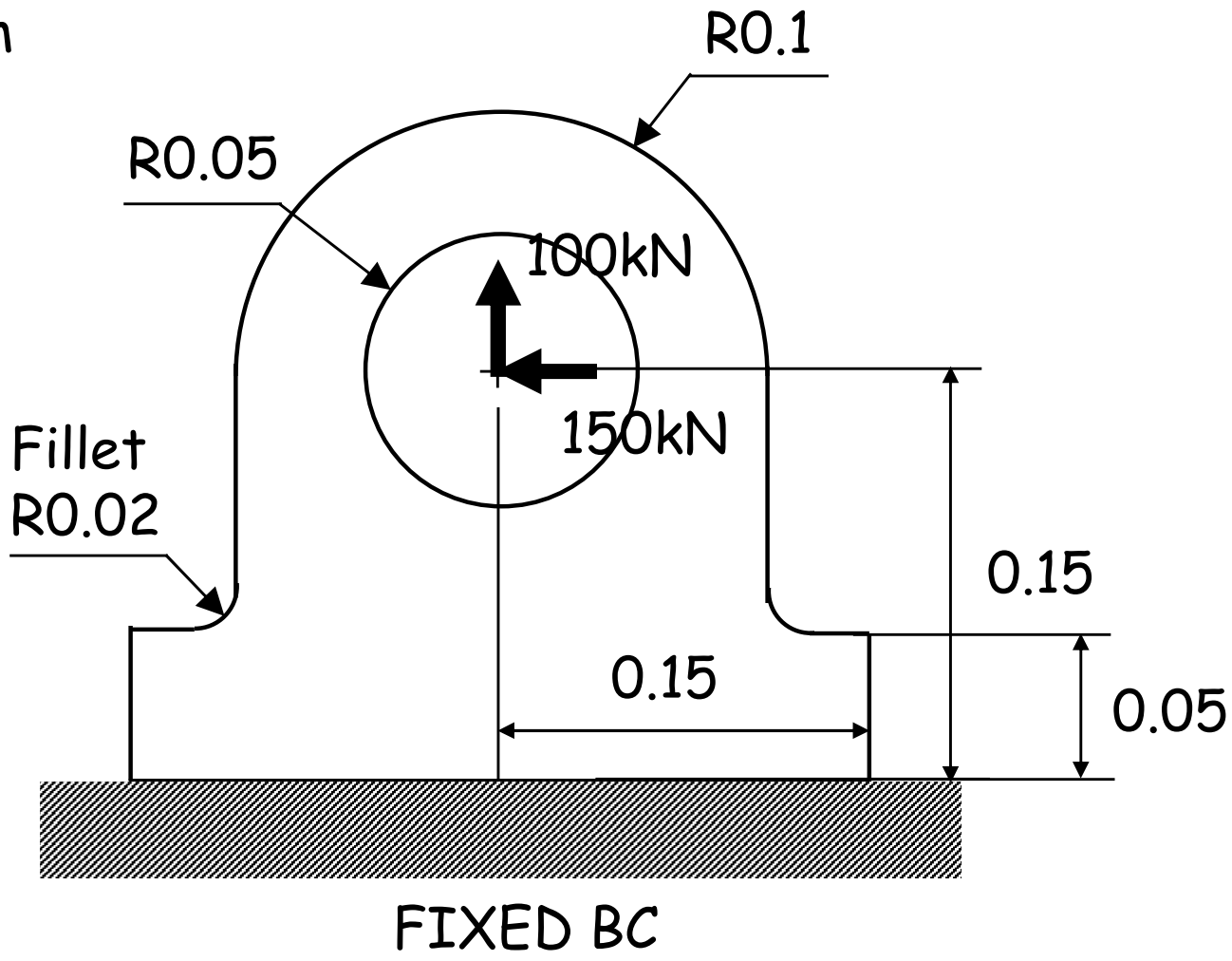


Convergence Study

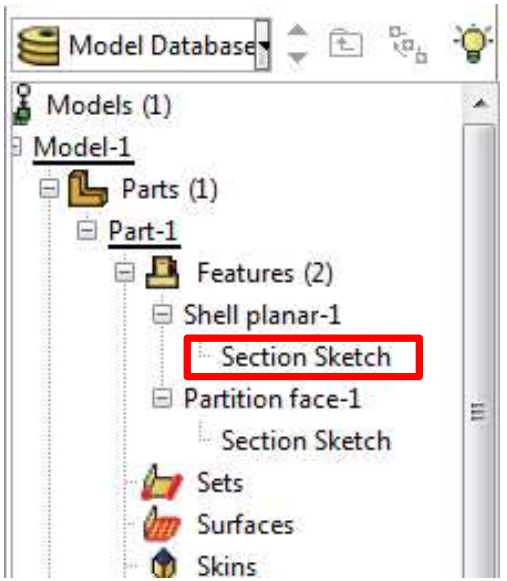


A BRACKET DESIGN

- A bracket design
  - Maximum stress is of 50MPa
  - Find optimum size of the outer radius (R_{out})
  - $0.07\ m \leq R_{out} \leq 0.1\ m$



PART MODULE



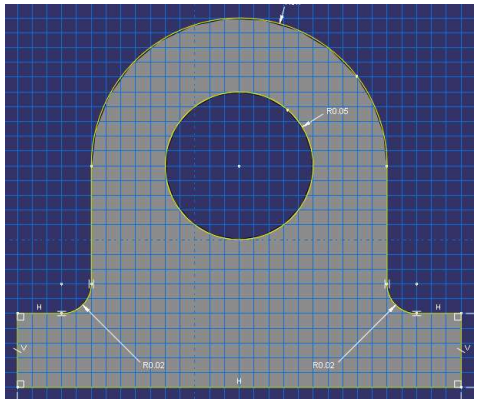
- Modify the current design
  - Modify part to modify the current design

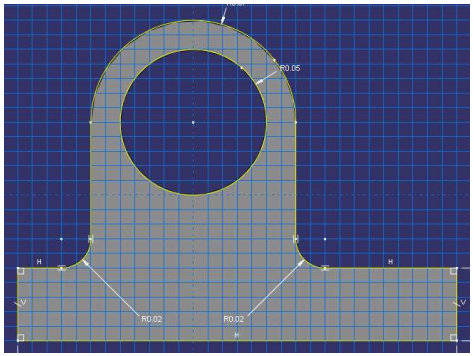
- Model tree / expand "Parts" /
  expand your part (default name: "Part-1") /
  expand "Features" /
  expand your sketch (default name: "Shell planar") /
  double click "Section Sketch"

- Menu/Edit/Dimension
  - Set the outer radius dimension to 0.07 as a trial and error process

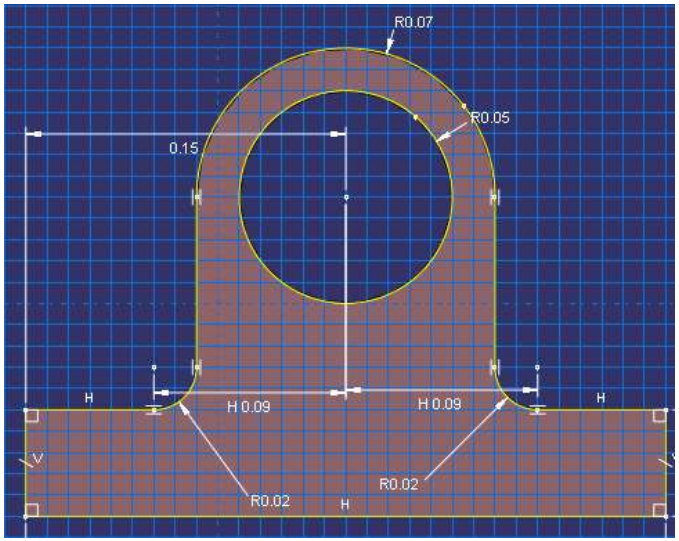
GEOMETRIC CONSTRAINTS

- What happens?
  - Un desirable design perturbation (unsymmetric design)





- Proper geometric constraints are needed
  - Add dimension from the shaft center and the edge at the bottom left to restrict the model remains symmetric as design parameter (the radius of the outer circle) is perturbed.



GEOMETRIC CONSTRAINTS

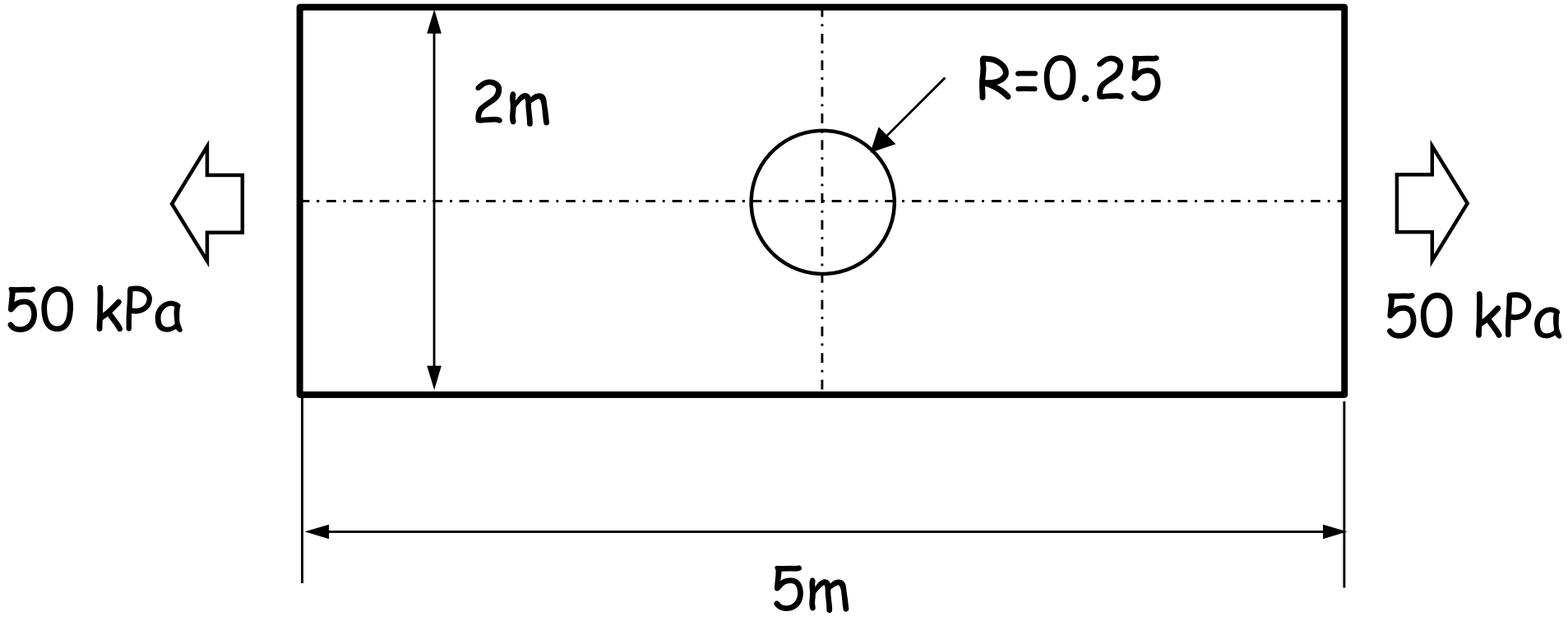
- After design perturbation
  - Max Von Mises stress increased to 69 Mpa
    (Violation of the maximum stress constraint)
  - Repeat the other trials to find optimum design while satisfying the maximum stress constraint

Tutorial 5-2:

2D Plane (basic modeling technique)

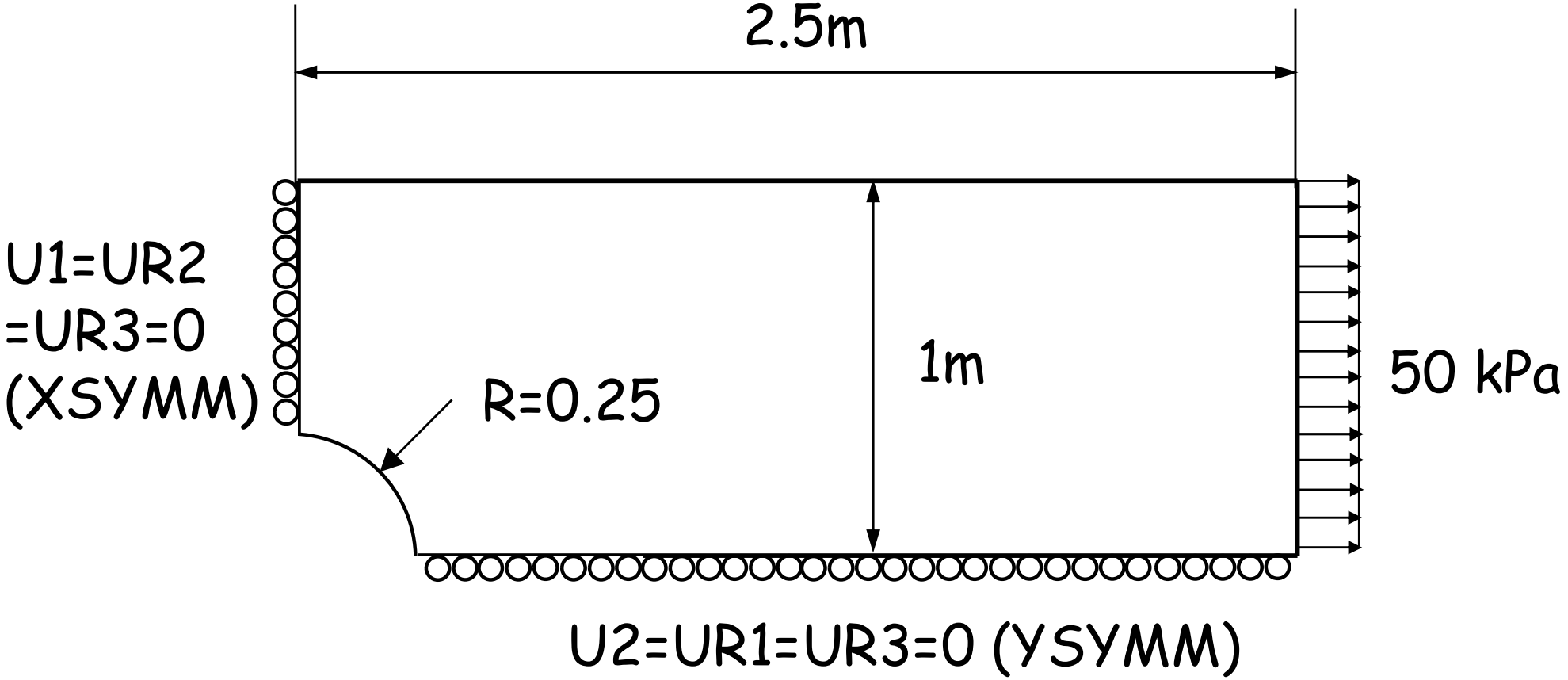
PANEL WITH A HOLE

- A panel with a hole
  - E = 200 GPa, ν=0.3
  - Thickness t = 0.01 m



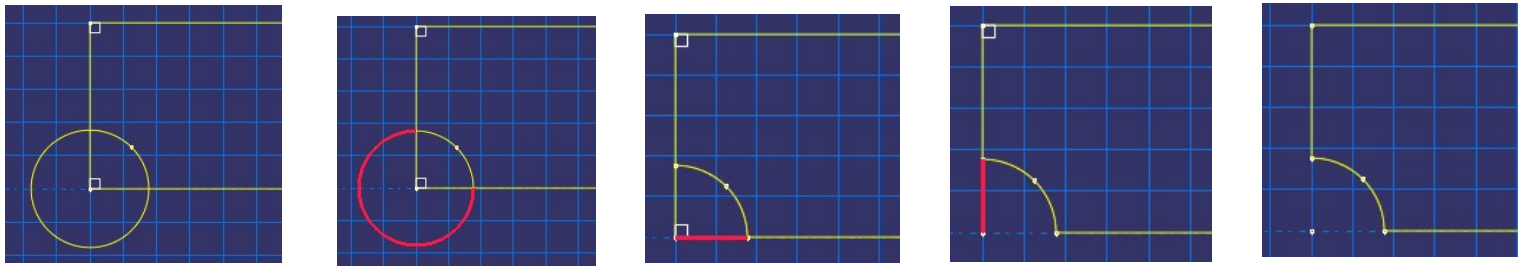
PANEL WITH A HOLE

- E = 200 GPa, ν=0.3
- Thickness t = 0.01 m



PARTS MODULE

- Parts
  - 2D Planar, Deformable, Shell, App Size = 10
  - Create lines (rectangle): (0, 0), (2.5, 1)
  - Create circle (center and perimeter): (0, 0), (0.25, 0)
  - Auto trim



- Tip
  - Even a circle has a starting point and ending point on circumference

PROPERTY / ASSEMBLY / STEPS MODULES

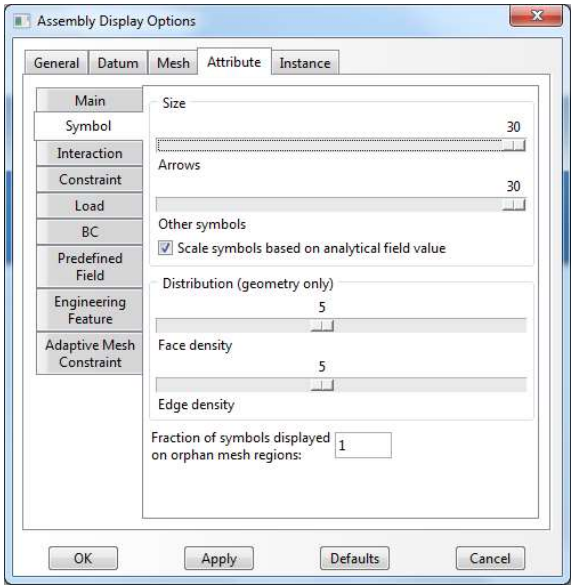
- Materials
  - Mechanical, Elasticity, Elastic
  - Young's modulus = 200E9, Poisson's ratio = 0.3
- Sections
  - Solid, Homogeneous
  - Set plane stress/strain thickness to 0.01 m
- Assign the section to the part
- Assembly, Instance
- Steps
  - Linear perturbation, Static

LOADS MODULE

- BCs
  - Step1, Symmetric, XSYMM and YSYMM

U1=UR2
=UR3=0
(XSYMM)

U2=UR1=UR3=0
(YSYMM)



- Tip
  - To change BC symbols: View, Assembly Display Options, Attribute
- Loads
  - Mechanical, Pressure, Uniform, -50000 (-50 kPa)

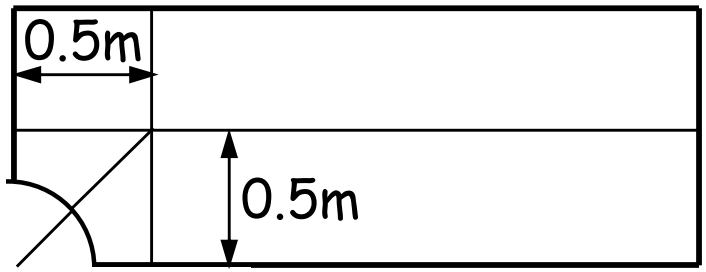
MESH MODULE (MESHING TECHNIQUE)

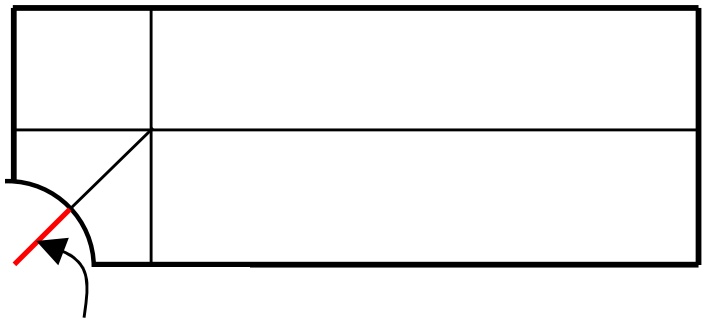
- Structured mesh
  - Follows predefined mesh patterns (Rule based meshing)
  - Predictable mesh shape
  - Not applicable for every geometry domain
    (If geometry domain is not affordable for structured mesh, a warning message and reasons will appear)
  - For 2D/Quad-dominated mesh, the geometry domain better have 4 edges

- Free mesh
  - No predefined mesh patterns
  - Flexibility
  - Impossible to predict a free mesh pattern

MESH MODULE (PARTITION FACE)

- Mesh
  - Menu: Tool/Partition/Partition Face/Sketch (sketch mode)
  - Draw 3 lines
  - Menu/Edit/Auto-trim, delete the red line



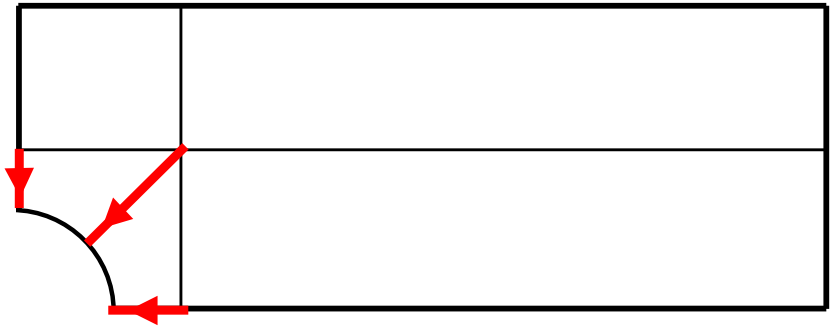


Auto trim

  - Assign Mesh Controls, Quad (Quad only), Structured
  - Global seed, size 0.1

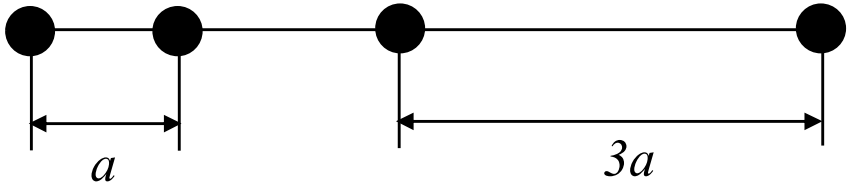
MESH MODULE (SEED MESH)

- Seed
  - Menu: Seed/Edge biased
  - Select lines, Bias ratio of 3, # of elements along the edge of 5



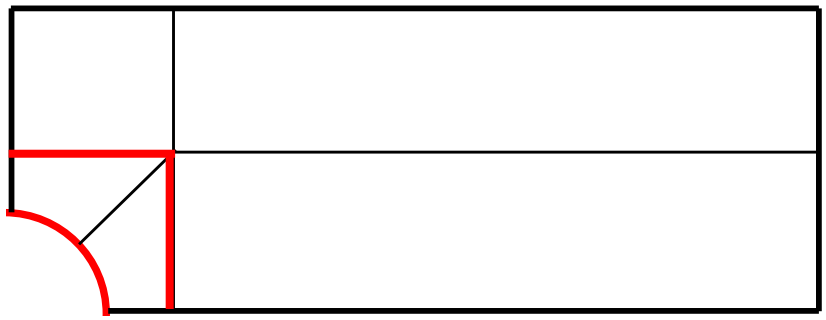
- Tip
  - When you select the edges, pick near the end where the mesh must be denser (red arrow shows mesh density direction)
  - Bias ratio: the ratio of size of a starting and ending element

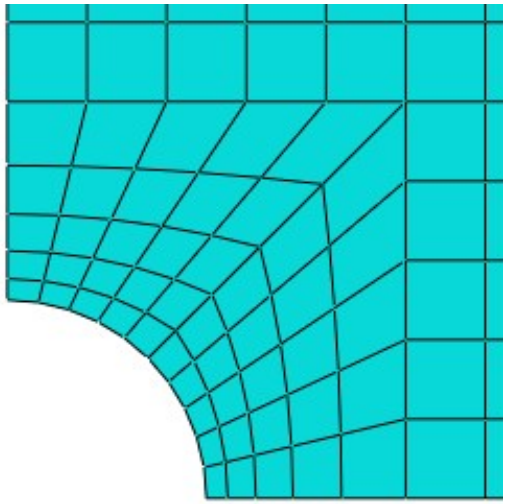
EX) Bias ratio of 3, # of elements along the edge of 3



MESH / JOB / VISUALIZATION MODULES

- Seed
  - Menu: Seed/Edge by number
  - Select lines, # of elements along the edge of 5





  - Mesn part
- Analysis, Create Job, Data Check, Submit
- Results
- Deformed plot, Stress plots
  - Field output, Mises