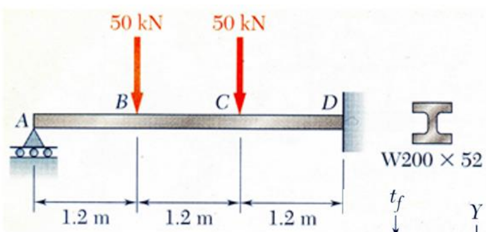
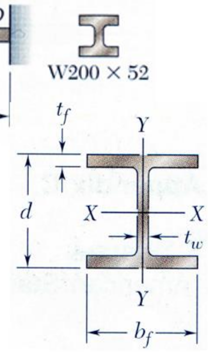
12/18/2018

Submit the compressed file as `(ID)_(name).zip` to [ftp://cdl.hanyang.ac.kr → CAE/Final_Lab] folder. It should contain the final results of each problem (equations and graphs) using PowerPoint `(ID.ppt)` and COMSOL files (`problem#-#.mph`).

1. [Beam deflection] For the beam and boundary conditions shown, solve the beam deflection at point B.
   1) Use "2D Beam" module. The number of elements should be 3. (10 pts)
   2) Use "MATLB" using the eliminated stiffness matrix(K) and load vector(f) of COMSOL result from "Assemble" option. (6 pts)

Analytic solution : $d_B = -3.19\,mm$

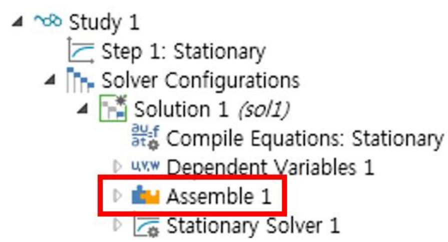




$E = 200\,GPa$

$d = 206\,mm, b_f = 204\,mm$

$t_f = 12.6\,mm, t_w = 7.9\,mm$

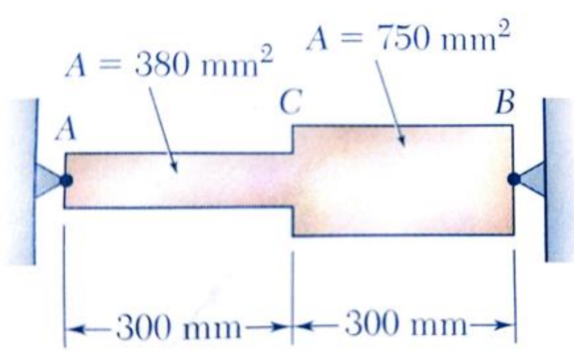



```
>> K\f

ans =

    0.0023
         0
   -0.0023
   -0.0012
         0
   -0.0032
   -0.0035
         0
```

$d_B = -0.032\ m$

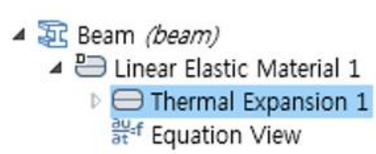
2. [Thermal expansion of structure] Determine the values of the stress at points A and B of steel bar as shown. When the temperature of the bar is -45 ˚C knowing that a close fit exists at both of the rigid supports when the temperature is +24 ˚C.

Analytic solution : $\sigma_A = 214.1$ MPa, $\sigma_B = 108.5$ MPa



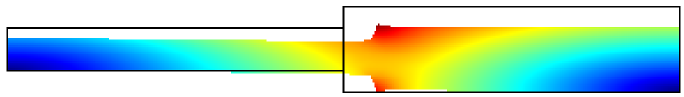
$E = 200$ GPa

$\alpha = 11.7 \times 10^{-6}\ /°C$

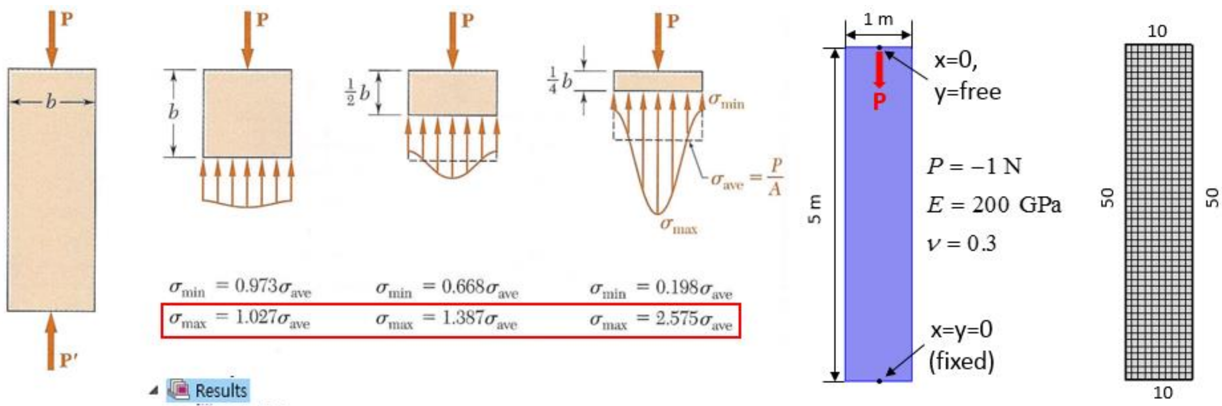


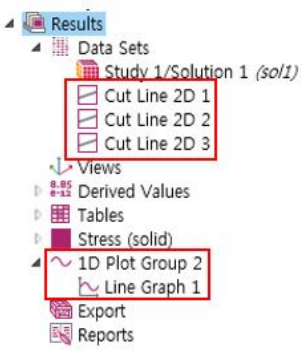
1) Use "2D Beam" module(`beam.smax`). (6 pts)

2) Use "2D Solid Mechanics" module(`solid.sx`). The bar thickness is 10 mm, and the widths in portions AC and CB are 38 mm and 75 mm, respectively. Set the proper boundary conditions which can display as below deformed shape (10 pts)



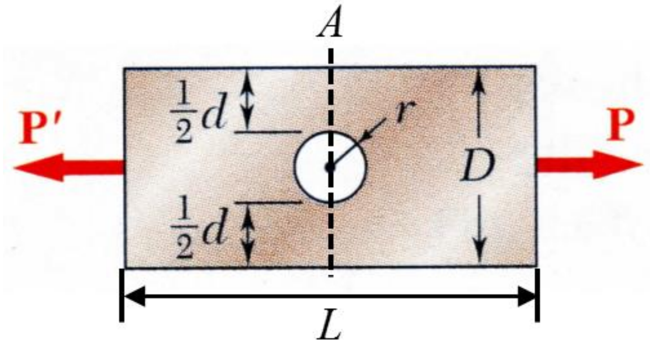
3. [Saint-Venant's principle] For the boundary conditions and mesh setting shown, plot the maximum normal stress(`solid.sy`) along the various cross sections of rectangular plate. Calculate the concentrated factors of each section and fill the table. (use "Cut Line 2D" and "1D Plot Group" options) (18 pts)





	Anal.	COMSOL
b	1.027	
1/2b	1.387	
1/4b	2.575	

4. [Stress concentration] The flat bar with a hole has a thickness of 1 m. The dimensions and boundary conditions are shown in the following figure.

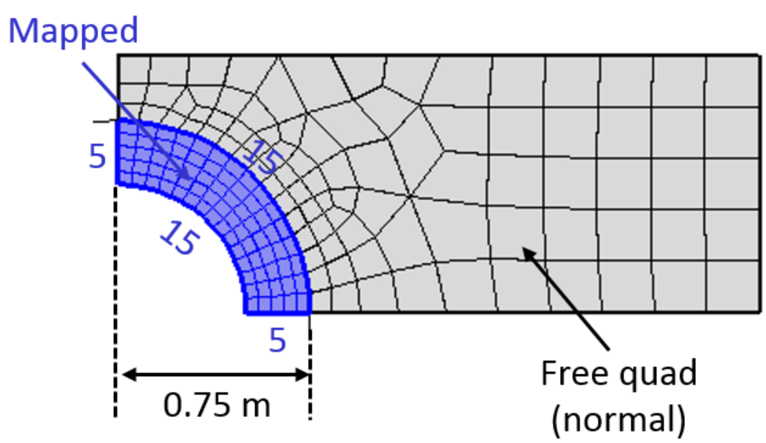


$E = 200$ GPa, $v = 0.3$

$L = 5$ m, $D = 2$ m, $P = 1$ Pa

$$\sigma_{ave} = \frac{P}{A} = \frac{P}{D-2r},\ K = \frac{\sigma_{max}}{\sigma_{ave}}$$

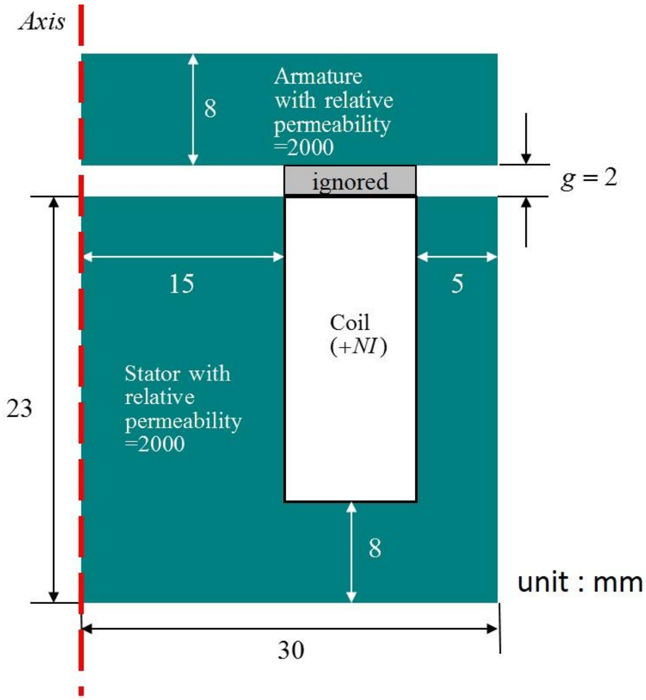
1) Compute the maximum normal stress(`solid.sx`) with r = 0.5 m. Construct a quarter model and set the mesh options as shown in the following figure. (15 pts)



2) Compute the value of K changing the radius(0.2 ~ 0.5 m) and fill the table. (8 pts)

r [m]	0.5	0.4	0.3	0.2
Anal.	2.18	2.30	2.46	2.62
COMSOL				

5. [Magnetic actuator] For the actuator and boundary conditions shown, assume that the magnetic energy density(`mf.Wm`) among the gap is constant and a part of gap between coil and armature is ignored. A proper air area(r: 50 m, z: 50 m) should be included. (use “Cut Line 2D” and “1D Plot Group” options)



$N = 2000$ turns

$I = 1\text{A}$

$$F = \frac{W_{den} V_{gap}}{g}$$

W_{den} : magnetic energy density [J/m^3]

V_{gap} : total volume on gap [m^3]

Analytic solution

$F = 240\,\text{N}$ (-z direction)

1) Plot the magnetic energy density(`mf.Wm`) along the center line of gap and compute the magnetic force on the gap. Know that V_{gap} is cylinder shape. (mesh option : Free Triangular(normal)) (20 pts)

2) Apply the mapped mesh(left side: 15x2, right side: 5x2) on the gap and compute the magnetic force.(set “Free Quad” with normal size on the other region) (7 pts)