M462-514 Project templates

This document reports the templates to guide the students toward their final project. We provide a couple project of options for both streams, Mech 462 (undergraduates) and Mech 514 graduates. The project marking is specified in a separate file ("M462-514_Project Report Template & Marking.pdf"). As indicated in the latter, the marking is based on the difficulty of the project, on the thoroughness of the execution of the calculations, and on the clarity of explanations. With this in mind, it allowed for 462 students to choose a project template that is reserved for 514, but 514 students cannot choose a 462 template.

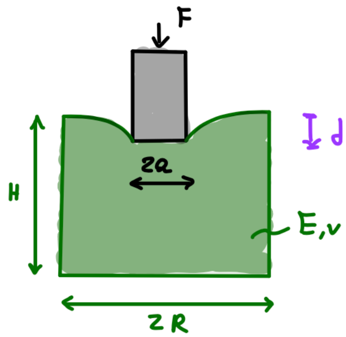
Each project involves first the solution of the problem via a custom-designed Matlab code, and then the solution via Abaqus. The comparison between Abaqus and Matlab may be used as first-order validation, however, a comparison with literature is required.

Under special circumstances, if requested by the student, we can consider a project template that is different from the ones specified, so long that is similar in nature.

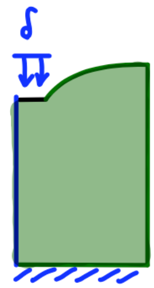
1) **Linear elastic *planar* indentation (plane strain or plane stress)**

[difficulty level 1]

Below the schematics the problem of planar flat-punch indentation. The problem should be first solved with Matlab, and then with Abaqus. 1st order validation via Matlab-Abaqus comparison and then comparison with the literature.

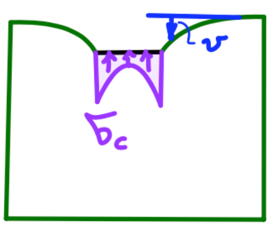


In green the linearly elastic specimen of modulus E and Poisson's ratio ν. Its height H and width R are much larger (at least 10-times) than the radius a of the rigid flat punch



You can take half model exploiting the symmetry.

Because the punch is rigid, you can assume prescribed displacements



Compare *displacements* and *stresses*.

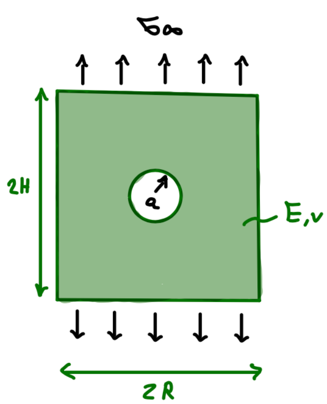
For validation, consult the references at the end of the document [1-2].

Nota Bene: To assume linear elasticity in the problem, you need to ensure that the average contact stress $\langle\sigma_c\rangle = \frac{1}{A_c}\int_{A_c}\sigma_c dA_c$ is much smaller than the elastic modulus, *i.e.* $\frac{\langle\sigma_c\rangle}{E} \le 0.1$

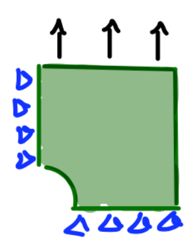
2) **Linear elastic stress concentration around a cylindrical hole in a plate (plane strain or plane stress)**

[difficulty level 2]

Below the schematics the problem of a cylindrical hole in a plate. The problem should be first solved with Matlab, and then with Abaqus. 1st order validation via Matlab-Abaqus comparison and then comparison with the literature.

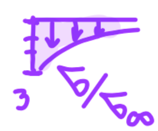


In green the linearly elastic specimen of modulus E and Poisson's ratio ν. Its height H and width R are much larger (at least 10-times) than the radius a of the hole



You can take a quarter of the plate exploiting the symmetry.

Because the hole is zero-stiffness, you can either give zero modulus to the elements in the hole, or construct a mesh around the hole.



Compare *displacements* and *stresses*.

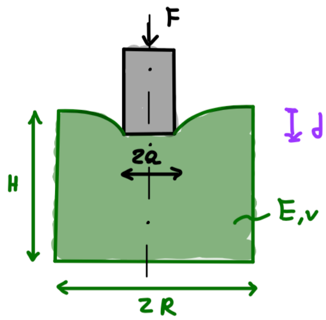
For validation, consult the references at the end of the document [2].

Nota Bene: To assume linear elasticity in the problem, you need to ensure that the remote stress σ_∞ is much smaller than the elastic modulus, *i.e.* $\frac{\sigma_\infty}{E} \le 0.1$

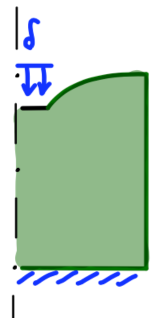
1) **Linear elastic *axisymmetric* indentation**

[difficulty level 1]

Below the schematics the problem of axisymmetric flat-punch indentation. The problem should be first solved with Matlab, and then with Abaqus. 1st order validation via Matlab-Abaqus comparison and then comparison with the literature.

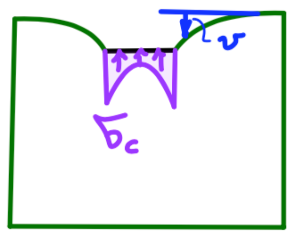


In green the linearly elastic specimen of modulus E and Poisson's ratio ν. Its height H and width R are much larger (at least 10-times) than the radius a of the rigid flat punch. The dash-point line is the axis of symmetry.



You can take a virtual slice of the model exploiting the *axial* symmetry. But be sure to use axisymmetric elements.

Because the punch is rigid, you can assume prescribed displacements



Compare *displacements* and *stresses*.

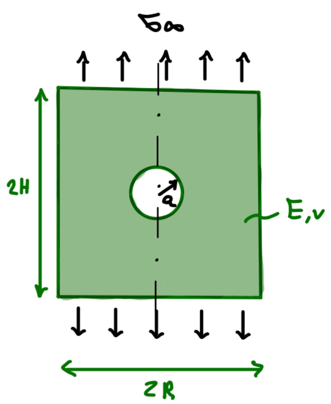
For validation, consult the references at the end of the document [1-2].

Nota Bene: To assume linear elasticity in the problem, you need to ensure that the average contact stress $\langle\sigma_c\rangle = \frac{1}{A_c}\int_{A_c}\sigma_c dA_c$ is much smaller than the elastic modulus, *i.e.* $\frac{\langle\sigma_c\rangle}{E} \leq 0.1$

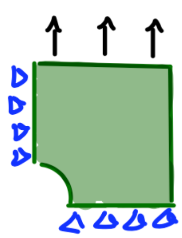
2) **Linear elastic stress concentration around a spherical cavity**

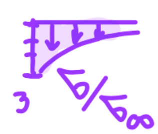
[difficulty level 2]

Below the schematics the problem of a spherical cavity in a 3D specimen. The problem should be first solved with Matlab, and then with Abaqus. 1st order validation via Matlab-Abaqus comparison and then comparison with the literature.



In green the linearly elastic specimen of modulus E and Poisson's ratio ν. Its height H and width R are much larger (at least 10-times) than the radius a of the hole. The dash-point line is the axis of symmetry.





You can take a virtual half-slice of the model by exploiting the axial symmetry. Be sure to use axisymmetric elements.

Because the hole is zero-stiffness, you can either give zero modulus to the elements in the hole, or construct a mesh around the hole.

Compare *displacements* and *stresses*.

For validation, consult the references at the end of the document [2].

Nota Bene: To assume linear elasticity in the problem, you need to ensure that the remote stress σ_∞ is much smaller than the elastic modulus, *i.e.* $\frac{\sigma_\infty}{E} \leq 0.1$

References:

[1] K. Johnson (1985). *Contact Mechanics*. Cambridge: Cambridge University Press. doi:10.1017/CBO9781139171731

[2] A.F. Bower (2009). *Applied Mechanics of Solids*. CRC Press. doi:10.1201/9781439802489