Exercise 3 Linear statics – analysis of finite element mesh density and element type influence (shape function type, integration, others)/analysis of solid or structural elements model (plate, beam) - isotropic linear elastic material

Based on Exercise 1:

- repeat the analysis from Exercise 1 and compute:
  - stress state in point D
  - strain stake in point F
  - displacement in point C
- repeat the analysis from Exercise 1 using different discretisation ('seed size' = 25,10,5 mm) for elements:
  - hexahedron (full and reduced integration)
  - tetrahedron (full and reduced integration)
  - linear and quadratic shape functions
  - how the stresses, strains or displacements are changing in A-F points?
- generate unsymmetrical FEM mesh and repeat the analysis from Exercise 1, what are the consequences?
- compare displacements in A, B and C with the results from any engineering software for beam model (Robot, Rm-Win, Soldis)
- compare stresses in point F with the results from any engineering software for beam model (Robot, Rm-Win, Soldis)
- compute reaction forces and compare them with the results from any engineering software for beam model (Robot, Rm-Win, Soldis)
- build beam model in Abaqus (based on Exercise 1) and compare it with 3D model - does the beam model make sense?
- build plane strain model in Abaqus (based on Exercise 1) and compare it with 3D model - does the plane strain model make sense?
• build plate model in Abaqus and compare it with 3D model - does the plate model make sense?
• using menu ‘Results options’ in bookmark ‘Computation’ check how ‘Averaging’ influences the contour plots e.g. reduced stresses (S > Mises) – compare contour maps by ‘Probe values’ tool
• create vertical displacement plots and refer them to the beam displacement animation

COMMENTS:
• it is possible to create many models in a single CAE file (Menu Tree tool)
• use ‘Probe values’ tool
• use 'View Cut' tool to analyse the model 'inside the volume'