LE10: Thick plate under pressure

LE10: Thick plate under pressure

This problem provides evidence that Abaqus can reproduce the result from the benchmark defined by NAFEMS and cited as the reference solution.

This page discusses:

- Elements tested
- Problem description
- Reference solution
- Results and discussion
- Input files

ProductsAbaqus/StandardAbaqus/Explicit

Elements tested

C3D20

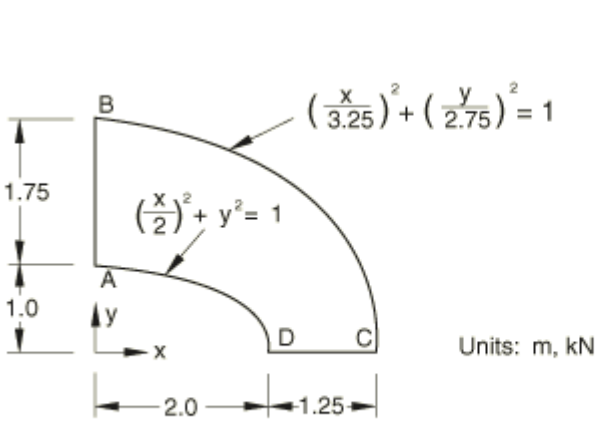
C3D20R

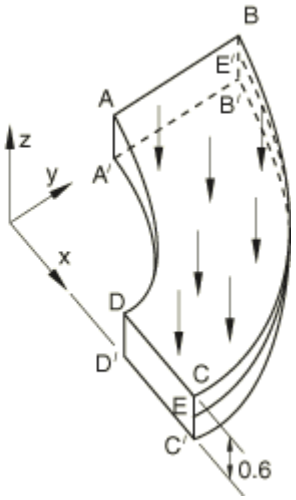
C3D10

C3D10HS

C3D10M

Problem description





Model:

Thick plate under uniform pressure.

Mesh:

A coarse and a fine mesh are tested.

Material:

Linear elastic, Young's modulus = 210 GPa, Poisson's ratio = 0.3, density = 7800 kg/m^3.

Boundary conditions:

uy= 0 on face DCD′C′. ux= 0 on face ABA′B′. ux=uy= 0 on face BCB′C′. uz= 0 on line EE′ (E is the midpoint of edge CC′; E′ is the midpoint of edge BB′).

Loading:

Uniform normal pressure of 1.0 MPa on the upper surface of the plate.

Reference solution

This is a test recommended by the National Agency for Finite Element Methods and Standards (U.K.): Test LE10 from NAFEMS Publication TNSB, Rev. 3, “The Standard NAFEMS Benchmarks,” October 1990.

Target solution: Direct stress, σyy= 5.38 MPa at point D.

Results and discussion

The Abaqus/Standard results are shown in Table 1. The values enclosed in parentheses are percentage differences with respect to the reference solution.

Table 1. Abaqus/Standard analysis.

Element	σyy, Coarse Mesh	σyy, Fine Mesh
C3D20	−6.72 MPa (+25.00%)	−5.64 MPa (+4.83%)
C3D20R	−7.93 MPa (+47.39%)	−5.53 MPa (+2.78%)
C3D10	−5.44 MPa (+1.15%)	−5.77 MPa (+7.24%)
C3D10HS	–5.08 MPa (−3.72%)	–5.51 MPa (+2.42%)
C3D10M	−5.57 MPa (+3.53%)	−5.89 MPa (+9.48%)

The C3D10 and C3D10M elements are more accurate with the coarse mesh than with the fine mesh: in the coarse meshes four elements come together at the point of interest, giving a more accurate result after averaging to the nodes. In the more refined mesh, only one element contains the point of interest; therefore, the extrapolation to the nodes is less accurate.

Unlike Abaqus/Standard, Abaqus/Explicit does not have the option for extrapolating integration point outputs (such as stresses) to the nodes. Consequently, the desired stress component at point D cannot be extracted except by rough interpretation of color contour plots. As an alternative, the value of σyy at an integration point near point D is compared between an Abaqus/Standard simulation and an Abaqus/Explicit simulation.

In the Abaqus/Explicit analyses the pressure is ramped up smoothly from zero to its final value of 1.0 MPa over a time period of 0.4 seconds, which is slow enough to be considered quasi-static (inertial effects play a minimal role).

Analysis Type	σyy, Coarse Mesh	σyy, Fine Mesh
Abaqus/Standard	−3.70 MPa	−4.61 MPa
Abaqus/Explicit	−3.79 MPa	−4.55 MPa

For the coarse mesh the point of comparison is at element 18, integration point 3. For the fine mesh the point of comparison is at element 199, integration point 1. Both are close neighbors of the physical corner point D.

Input files

Abaqus/Standard input files

Coarse mesh tests:

nle10fkc.inp

C3D20 elements.

nle10rkc.inp

C3D20R elements.

nle10c_c3d10.inp

C3D10 elements.

nle10c_c3d10hs.inp

C3D10HS elements.

nle10c_c3d10m.inp

C3D10M elements.

Fine mesh tests:

nle10fkf.inp

C3D20 elements.

nle10rkf.inp

C3D20R elements.

nle10f_c3d10.inp

C3D10 elements.

nle10f_c3d10hs.inp

C3D10HS elements.

nle10f_c3d10m.inp

C3D10M elements.

Abaqus/Explicit input files

exxle10_c.inp

C3D10M elements, coarse mesh.

exxle10_f.inp

C3D10M elements, fine mesh.