cFEA verification report

EngCoSuite, cloud Finite Element Analysis. Solver code_aster 17.4.0, mesher Gmsh 4.13.1. Report date 2026-11-18.

Summary

Report date	2026-11-18
Solver	code_aster 17.4.0 (open source, EDF)
Mesher	Gmsh 4.13.1 (open source)
Verification cases	19
Checks	61
Result	**All 61 checks passed**

What was checked, and how

Each verification case is a small model whose answer is known from a hand calculation or from textbook theory (beam, plate and heat-conduction formulas). The case is run through the same path a customer's model takes: the geometry is read, meshed with Gmsh, solved with code_aster, and the results are processed by cFEA. The value cFEA reports is then compared with the reference value, and the check passes only if the difference is inside the stated tolerance.

The whole suite is re-run after every change to cFEA, the solver or the mesher, and before every release, so this report describes the version named above. The tolerances are not zero: a finite element model approaches the exact answer as the mesh is refined, and the elements used here have known limits (for example, thin-plate shell elements do not include transverse shear). Each tolerance is set from what the element and mesh size should achieve, and the difference found is shown beside it.

What this report does and does not show

- It shows that, for these problems, cFEA solves the equations correctly and reports the right quantities in the right units.
- It does not replace engineering judgement on a customer's own model: the mesh, the supports, the loads and the material data entered are the responsibility of the person who makes and issues the analysis. Mesh refinement and a check of the result against a hand calculation remain good practice.
- cFEA covers linear elastic analysis (static and natural frequencies) and steady-state heat conduction, on one solid part or on thin shells. It does not cover contact, assemblies, plasticity, large deformation or thermal-structural coupling.
- This is verification (is the program solving the equations right?). It is not an external certification.

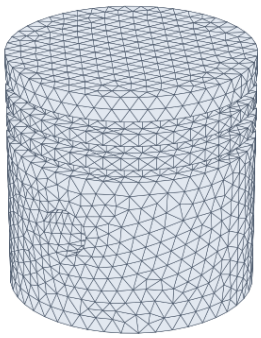
Geometry import

A STEP file whose edges are 0.01 mm off their faces still meshes (healed on import)

A DN100 piston exported from FreeCAD 1.1. Its Pad and Pocket features touch the skirt, and OpenCascade's check reports BOPAlgo_InvalidCurveOnSurface on 4 edges. Without the repair on import Gmsh fails at every element size with 'No elements in volume 1' or 'PLC Error: two facets intersect'. The runner heals the shape within 1e-4 of its diagonal (geometry._heal) and the mesh is made. The solid is 237,811 mm3 with 27 faces.

Geometry	A STEP file (piston.step)
Mesh	element size 6.9 mm, quadratic elements

Geometry, supports and loads



6054 surface triangles, 3025 surface nodes

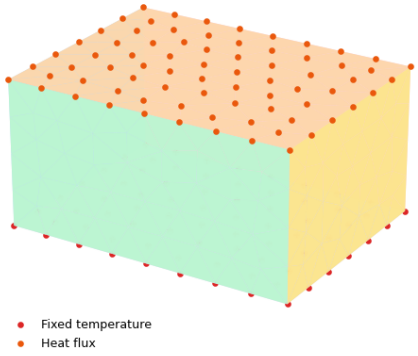
Mesh

Quantity	Reference	cFEA	Difference	Allowed	Result
Faces found in the file Reference: the 27 faces of the Body, none lost by the repair	27	27	0	+/- 0	**Pass**
Volume of the part Reference: the volume FreeCAD reports for the Body	237,811 mm3	237,811 mm3	0.00 %	0.01 %	**Pass**
Inverted elements in the mesh	0	0	0	+/- 0	**Pass**

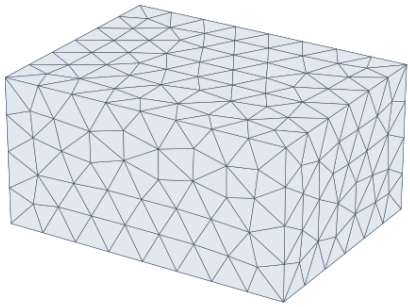
Block from an STL file (surface patches, steady-state thermal)

An STL has no CAD faces: the triangles are grouped into patches at sharp edges. 40 x 30 x 20 mm block, 5000 W/m2 through 20 mm of steel.

Geometry	Box 40 x 30 x 20 mm (built in the test, exported to STL)
Analysis	Steady-state thermal, solid elements
Material	Steel: conductivity 50 W/(m.K)
Mesh	element size 5 mm, quadratic elements
Support	Fixed temperature on the faces or edges in the plane Z = 0 mm (T (degC) = 20)
Load	Heat flux on the faces or edges in the plane Z = 20 mm (q (W/m2) = 5000)

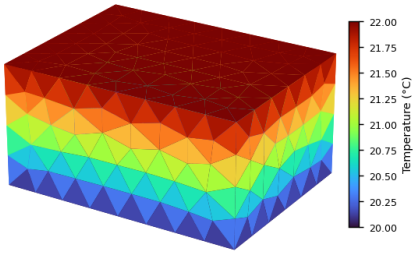


Geometry, supports and loads



534 surface triangles, 269 surface nodes

Mesh



Main result

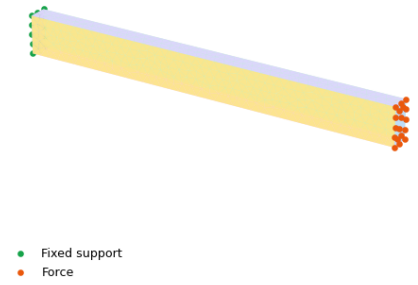
Quantity	Reference	cFEA	Difference	Allowed	Result
Faces found in the file Reference: six patches found from the triangles	6	6	0	+/- 0	**Pass**
Maximum temperature Reference: T0 + q L / k	22 degC	22 degC	+0.00 %	0.1 %	**Pass**

Linear static analysis

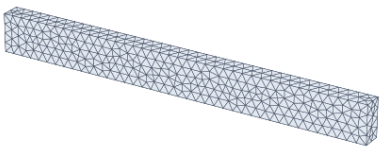
Cantilever beam, end load (linear static)

Steel bar 1000 x 50 x 100 mm fixed at one end, 1000 N at the other. Beam theory with the shear term.

Geometry	Box 1000 x 50 x 100 mm
Analysis	Linear static, solid elements
Material	Steel: E = 210 GPa, Poisson's ratio 0.3, density 7850 kg/m3
Mesh	element size 25 mm, quadratic elements
Support	Fixed support on the faces or edges in the plane X = 0 mm
Load	Force on the faces or edges in the plane X = 1000 mm (FZ (N) = -1000)

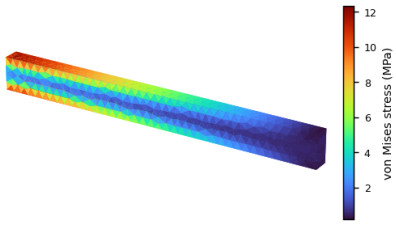


Geometry, supports and loads



1224 surface triangles, 614 surface nodes

Mesh



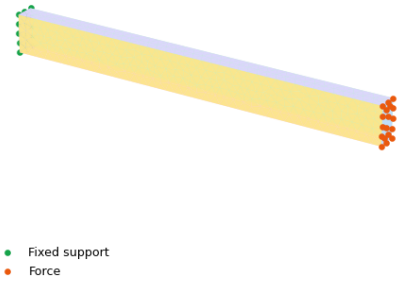
Main result

Quantity	Reference	cFEA	Difference	Allowed	Result
Maximum displacement Reference: PL^3/(3EI) + PL/(kGA), I = 4.1667e6 mm4, k = 5/6	0.3839 mm	0.383 mm	-0.25 %	2 %	**Pass**
Reaction force, Z Reference: reaction = applied load	1,000 N	1,000 N	0 N	+/- 0.01 N	**Pass**
Strain energy Reference: P^2 L^3/(6EI) + P^2 L/(2kGA)	0.192 J	0.1909 J	-0.54 %	2 %	**Pass**
Mass Reference: rho x volume	39.25 kg	39.25 kg	0.00 %	0.01 %	**Pass**

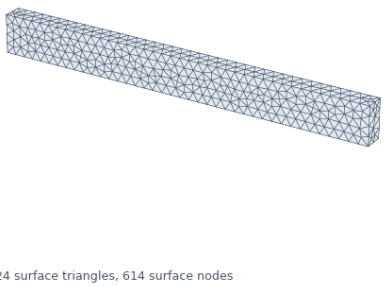
Cantilever beam solved iteratively (PETSc multigrid)

The cantilever case forced onto the iterative solver: the same hand calculation must hold, and the run must say it was iterative.

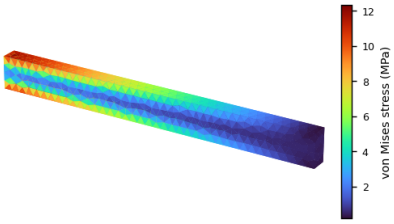
Geometry	Box 1000 x 50 x 100 mm
Analysis	Linear static, solid elements
Material	Steel: E = 210 GPa, Poisson's ratio 0.3, density 7850 kg/m3
Mesh	element size 25 mm, quadratic elements
Support	Fixed support on the faces or edges in the plane X = 0 mm
Load	Force on the faces or edges in the plane X = 1000 mm (FZ (N) = -1000)



Geometry, supports and loads



Mesh



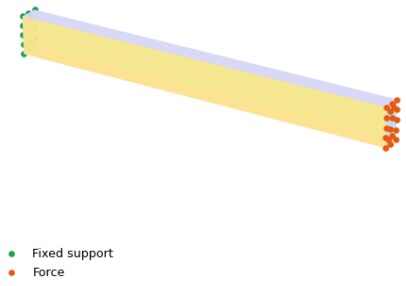
Main result

Quantity	Reference	cFEA	Difference	Allowed	Result
Maximum displacement Reference: PL^3/(3EI) + PL/(kGA), I = 4.1667e6 mm4, k = 5/6	0.3839 mm	0.383 mm	-0.25 %	2 %	**Pass**
Reaction force, Z Reference: reaction = applied load	1,000 N	1,000 N	0 N	+/- 0.01 N	**Pass**
Strain energy Reference: P^2 L^3/(6EI) + P^2 L/(2kGA)	0.192 J	0.1909 J	-0.54 %	2 %	**Pass**
Mass Reference: rho x volume	39.25 kg	39.25 kg	0.00 %	0.01 %	**Pass**
Iterative solver used (1 = yes) Reference: the solver used was the iterative one	1	1	0	+/- 0	**Pass**
Fell back to the direct solver (1 = yes) Reference: and it did not fall back	0	0	0	+/- 0	**Pass**

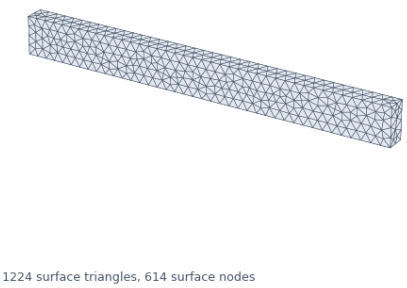
Iterative solve that cannot converge falls back to the direct solver

The iteration limit is 1, so the iterative solve fails; the run must repeat with the direct solver, give the same answers, and say so.

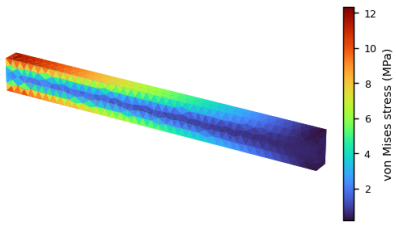
Geometry	Box 1000 x 50 x 100 mm
Analysis	Linear static, solid elements
Material	Steel: E = 210 GPa, Poisson's ratio 0.3, density 7850 kg/m3
Mesh	element size 25 mm, quadratic elements
Support	Fixed support on the faces or edges in the plane X = 0 mm
Load	Force on the faces or edges in the plane X = 1000 mm (FZ (N) = -1000)



Geometry, supports and loads



Mesh



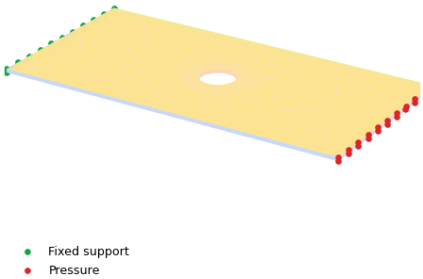
Main result

Quantity	Reference	cFEA	Difference	Allowed	Result
Maximum displacement Reference: PL^3/(3EI) + PL/(kGA), I = 4.1667e6 mm4, k = 5/6	0.3839 mm	0.383 mm	-0.25 %	2 %	**Pass**
Reaction force, Z Reference: reaction = applied load	1,000 N	1,000 N	0 N	+/- 0.01 N	**Pass**
Strain energy Reference: P^2 L^3/(6EI) + P^2 L/(2kGA)	0.192 J	0.1909 J	-0.54 %	2 %	**Pass**
Mass Reference: rho x volume	39.25 kg	39.25 kg	0.00 %	0.01 %	**Pass**
Iterative solver used (1 = yes) Reference: the answer came from the direct solver	0	0	0	+/- 0	**Pass**
Fell back to the direct solver (1 = yes) Reference: after the iterative solver failed	1	1	0	+/- 0	**Pass**

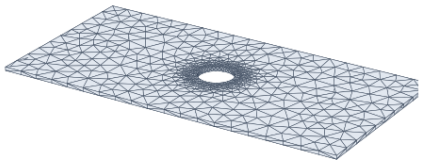
Plate with a hole in tension, from a STEP file (stress concentration)

200 x 100 x 2 mm plate (thin: plane stress), 20 mm hole, 100 MPa tension. Peterson: Kt (net section) = 3 - 3.14 r + 3.667 r^2 - 1.527 r^3 with r = d/w = 0.2. A thick plate reads higher at mid-thickness and lower at the faces (measured: 333 and 308 MPa at 10 mm), which the plane-stress handbook value does not describe.

Geometry	Box 200 x 100 x 2 mm, with a hole (built in the test, exported to STEP)
Analysis	Linear static, solid elements
Material	Steel: E = 210 GPa, Poisson's ratio 0.3, density 7850 kg/m3
Mesh	element size 10 mm, quadratic elements, 1 local refinement(s)
Support	Fixed support on the faces or edges in the plane X = 0 mm
Load	Pressure on the faces or edges in the plane X = 200 mm (p (Pa) = -1e+08)

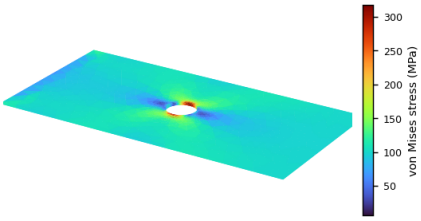


Geometry, supports and loads



2400 surface triangles, 1200 surface nodes

Mesh



Main result

Quantity	Reference	cFEA	Difference	Allowed	Result
Faces found in the file Reference: six plate faces + the hole	7	7	0	+/- 0	**Pass**
Volume of the part Reference: 200 x 100 x 2 - pi 10^2 x 2	39,372 mm3	39,372 mm3	+0.00 %	0.01 %	**Pass**
Maximum principal stress Reference: Kt x sigma / (1 - d/w) = 2.5065 x 100 / 0.8 MPa	313.3 MPa	319.9 MPa	+2.09 %	5 %	**Pass**
Reaction force, X Reference: 100 MPa x 200 mm2	-20,000 N	-20,000 N	0 N	+/- 0.5 N	**Pass**
Maximum von Mises stress (element value) Reference: the largest element's own von Mises value (measured 294.3 MPa): below the smoothed nodal peak of 317 MPa	294.3 MPa	294.3 MPa	+0.00 %	5 %	**Pass**
Largest stress difference (max - min principal) Reference: largest gap between the smoothed nodal stress and the elements' own (measured 44.2 MPa)	44.19 MPa	44.19 MPa	+0.00 %	25 %	**Pass**

Behaviour checked	Result
Mesh is finer inside the refined region element size inside / outside = 0.264 (limit 0.5)	**Pass**

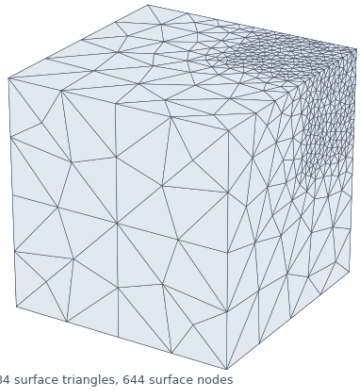
Mesh controls and refusals

Mesh control: refine inside a box

A 100 mm cube meshed at 25 mm with one refinement; the surface elements in the refined region must be clearly smaller.

Geometry	Box 100 x 100 x 100 mm
Mesh	element size 25 mm, linear elements, 1 local refinement(s)

Geometry, supports and loads



Mesh

Quantity	Reference	cFEA	Difference	Allowed	Result
Inverted elements in the mesh	0	0	0	+/- 0	**Pass**

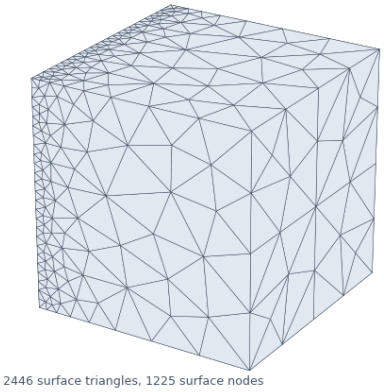
Behaviour checked	Result
Mesh is finer inside the refined region element size inside / outside = 0.301 (limit 0.4)	**Pass**

Mesh control: refine on a face

A 100 mm cube meshed at 25 mm with one refinement; the surface elements in the refined region must be clearly smaller.

Geometry	Box 100 x 100 x 100 mm
Mesh	element size 25 mm, linear elements, 1 local refinement(s)

Geometry, supports and loads



Mesh

Quantity	Reference	cFEA	Difference	Allowed	Result
Inverted elements in the mesh	0	0	0	+/- 0	**Pass**

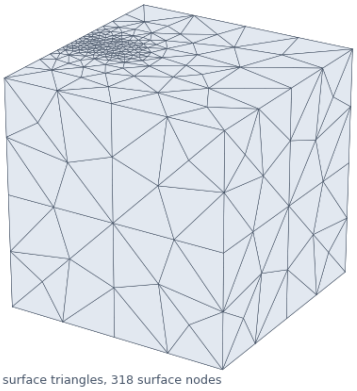
Behaviour checked	Result
Mesh is finer inside the refined region element size inside / outside = 0.359 (limit 0.4)	**Pass**

Mesh control: refine where painted

A 100 mm cube meshed at 25 mm with one refinement; the surface elements in the refined region must be clearly smaller.

Geometry	Box 100 x 100 x 100 mm
Mesh	element size 25 mm, linear elements, 1 local refinement(s)

Geometry, supports and loads



632 surface triangles, 318 surface nodes

Mesh

Quantity	Reference	cFEA	Difference	Allowed	Result
Inverted elements in the mesh	0	0	0	+/- 0	**Pass**

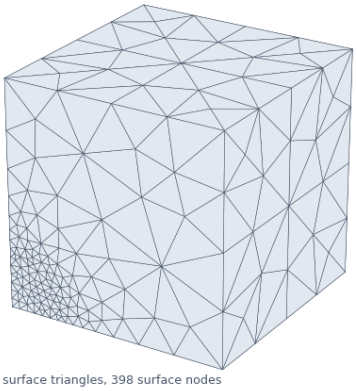
Behaviour checked	Result
Mesh is finer inside the refined region element size inside / outside = 0.207 (limit 0.4)	**Pass**

Mesh control: refine inside a sphere

A 100 mm cube meshed at 25 mm with one refinement; the surface elements in the refined region must be clearly smaller.

Geometry	Box 100 x 100 x 100 mm
Mesh	element size 25 mm, linear elements, 1 local refinement(s)

Geometry, supports and loads



792 surface triangles, 398 surface nodes

Mesh

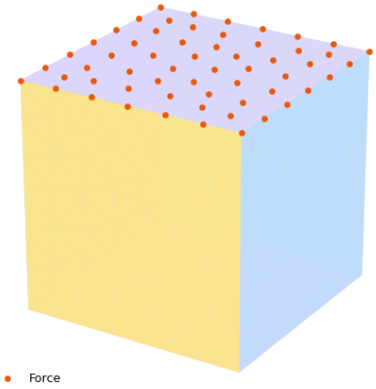
Quantity	Reference	cFEA	Difference	Allowed	Result
Inverted elements in the mesh	0	0	0	+/- 0	**Pass**

Behaviour checked	Result
Mesh is finer inside the refined region element size inside / outside = 0.246 (limit 0.4)	**Pass**

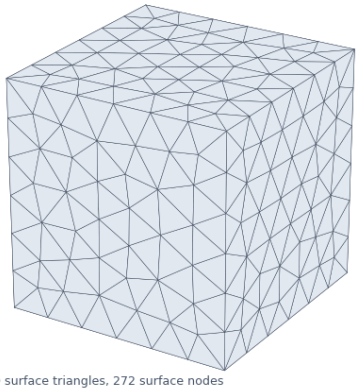
A static run without supports is refused

The part would move freely; the runner must say so instead of failing in the solver.

Geometry	Box 100 x 100 x 100 mm
Analysis	Linear static, solid elements
Material	Steel: E = 210 GPa, Poisson's ratio 0.3, density 7850 kg/m3
Mesh	element size 25 mm, linear elements
Load	Force on the faces or edges in the plane Z = 100 mm (FZ (N) = -100)



Geometry, supports and loads



Mesh

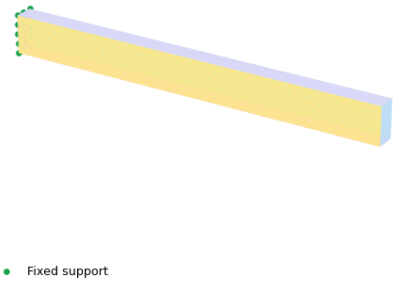
Behaviour checked	Result
The run is refused with a message No supports are defined: the part would move freely.	**Pass**

Natural frequencies (modal analysis)

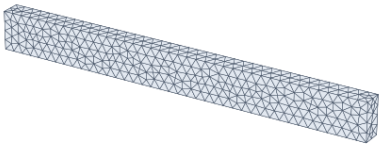
Cantilever beam, natural frequencies (modal)

The same bar; Euler-Bernoulli bending frequencies f = (lambda^2 / 2 pi L^2) sqrt(EI / rho A).

Geometry	Box 1000 x 50 x 100 mm
Analysis	Natural frequencies (modal), solid elements
Material	Steel: E = 210 GPa, Poisson's ratio 0.3, density 7850 kg/m3
Mesh	element size 25 mm, quadratic elements
Support	Fixed support on the faces or edges in the plane X = 0 mm



Geometry, supports and loads



Mesh

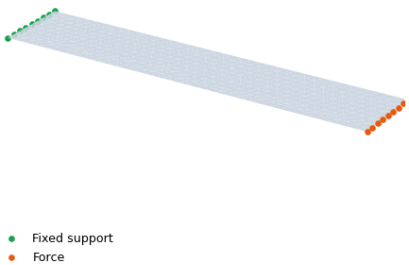
Quantity	Reference	cFEA	Difference	Allowed	Result
Natural frequency, mode 1 Reference: lambda = 1.8751, bending about the weak axis	41.77 Hz	41.92 Hz	+0.35 %	1 %	**Pass**
Natural frequency, mode 2 Reference: lambda = 1.8751, bending about the strong axis	83.54 Hz	83.19 Hz	-0.42 %	1 %	**Pass**
Natural frequency, mode 3 Reference: lambda = 4.6941, second weak-axis mode	261.8 Hz	259.7 Hz	-0.79 %	2 %	**Pass**

Shell elements (thin plates)

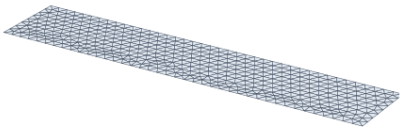
Plate cantilever, line load on the free edge (shell elements, linear static)

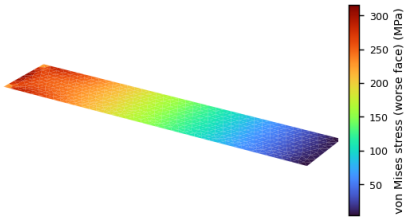
Steel strip 1000 x 200 x 10 mm modelled as a DKT shell, clamped along one short edge, 1000 N spread along the opposite edge. Euler-Bernoulli beam theory (the shell has no shear deformation).

Geometry	Flat plate (a surface) 1000 x 200 mm
Analysis	Linear static, shell elements (thin plates)
Shell thickness	10 mm
Material	Steel: E = 210 GPa, Poisson's ratio 0.3, density 7850 kg/m3
Mesh	element size 25 mm, linear elements
Support	Fixed support on the faces or edges in the plane X = 0 mm
Load	Force on the faces or edges in the plane X = 1000 mm (FZ (N) = -1000)



Geometry, supports and loads





Main result

800 surface triangles, 449 surface nodes

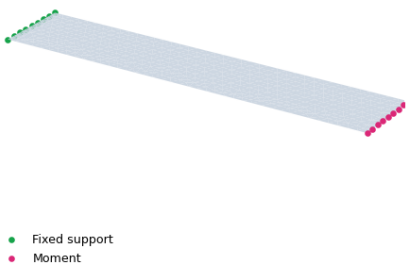
Mesh

Quantity	Reference	cFEA	Difference	Allowed	Result
Maximum displacement Reference: P L^3 / (3 E I), I = 200 x 10^3 / 12 = 16667 mm4 (the plate effect, nu^2, adds a little)	95.24 mm	93.32 mm	-2.02 %	4 %	**Pass**
Reaction force, Z Reference: reaction = applied load	1,000 N	1,000 N	0 N	+/- 0.01 N	**Pass**
Maximum von Mises stress Reference: 6 P L / (b t^2) = 6 x 1000 x 1000 / (200 x 10^2) = 300 MPa at the root (the clamped Poisson restraint adds a few percent at the corners)	300 MPa	315.4 MPa	+5.12 %	8 %	**Pass**

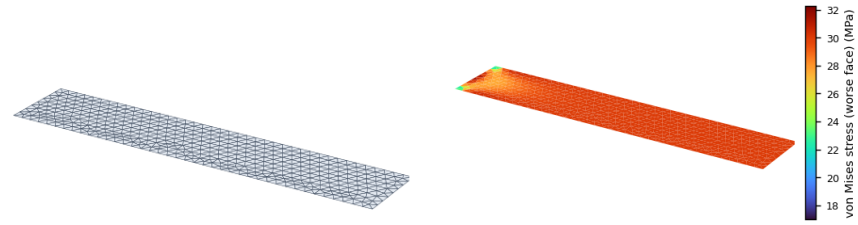
Plate cantilever, moment on the free edge (shell elements, linear static)

Steel strip 1000 x 200 x 10 mm modelled as a DKT shell, clamped along one short edge, a 100 N.m moment about Y applied along the opposite edge (spread over the edge nodes as nodal moments). Euler-Bernoulli beam theory.

Geometry	Flat plate (a surface) 1000 x 200 mm
Analysis	Linear static, shell elements (thin plates)
Shell thickness	10 mm
Material	Steel: E = 210 GPa, Poisson's ratio 0.3, density 7850 kg/m3
Mesh	element size 25 mm, linear elements
Support	Fixed support on the faces or edges in the plane X = 0 mm
Load	Moment on the faces or edges in the plane X = 1000 mm (MY (N.m) = 100)



Geometry, supports and loads



Main result

800 surface triangles, 449 surface nodes

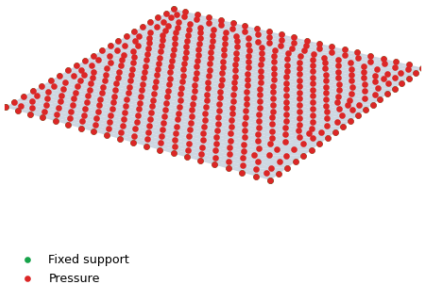
Mesh

Quantity	Reference	cFEA	Difference	Allowed	Result
Maximum displacement Reference: M L^2 / (2 E I), M = 1e5 N.mm, L = 1000 mm, I = 200 x 10^3 / 12 = 16667 mm4	14.29 mm	14.1 mm	-1.29 %	4 %	**Pass**
Maximum rotation Reference: M L / (E I)	0.02857 rad	0.02837 rad	-0.70 %	4 %	**Pass**
Maximum von Mises stress Reference: 6 M / (b t^2) = 6 x 1e5 / (200 x 10^2) = 30 MPa (uniform along the strip; the clamped Poisson restraint adds a few percent at the root); the same node-averaged corner excess as the edge-load case	30 MPa	32.26 MPa	+7.54 %	10 %	**Pass**
Applied moment about Y Reference: the applied moment about Y	100 N.m	100 N.m	0 N.m	+/- 0.001 N.m	**Pass**

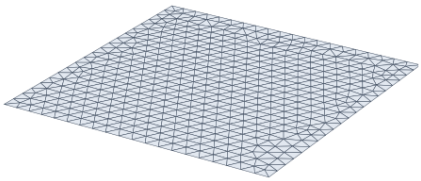
Square plate clamped on all four edges, uniform pressure (shell elements, linear static)

Steel plate 1000 x 1000 x 10 mm modelled as a DKT shell, clamped along its four edges, 0.01 MPa on the face. Plate theory (Timoshenko, clamped square plate): centre deflection 0.00126 q a^4 / D, clamped-edge moment 0.0513 q a^2 per unit length.

Geometry	Flat plate (a surface) 1000 x 1000 mm
Analysis	Linear static, shell elements (thin plates)
Shell thickness	10 mm
Material	Steel: E = 210 GPa, Poisson's ratio 0.3, density 7850 kg/m3
Mesh	element size 50 mm, linear elements
Support	Fixed support on the faces or edges in the plane X = 0 mm
Support	Fixed support on the faces or edges in the plane X = 1000 mm
Support	Fixed support on the faces or edges in the plane Y = 0 mm
Support	Fixed support on the faces or edges in the plane Y = 1000 mm
Load	Pressure on face 1 (p (Pa) = 10000)

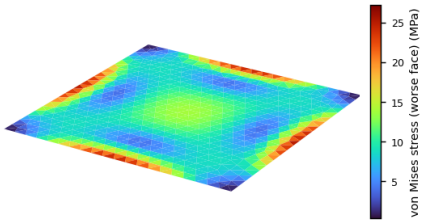


Geometry, supports and loads



940 surface triangles, 511 surface nodes

Mesh



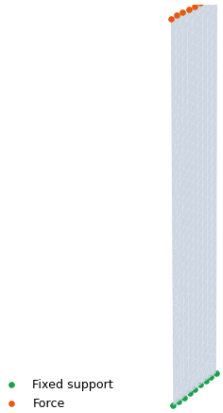
Main result

Quantity	Reference	cFEA	Difference	Allowed	Result
Maximum displacement Reference: 0.00126 q a^4 / D, D = E t^3 / (12 (1 - nu^2)) = 1.923e7 N.mm	0.6552 mm	0.6587 mm	+0.53 %	3 %	**Pass**
Reaction force, Z Reference: reaction = q a^2 = 0.01 x 1000^2	10,000 N	10,000 N	0 N	+/- 1 N	**Pass**
Maximum von Mises stress Reference: 6 M / t^2 x sqrt(1 - nu + nu^2): at a clamped edge the stress along the edge is nu times the one across it; M = 0.0513 q a^2 = 513 N.mm/mm	27.36 MPa	27.2 MPa	-0.57 %	4 %	**Pass**
Mass Reference: rho x area x t	78.5 kg	78.5 kg	+0.00 %	0.01 %	**Pass**

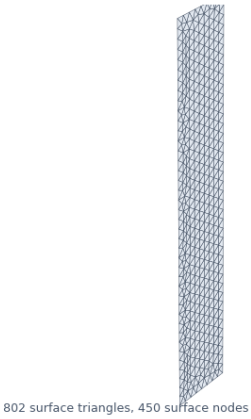
Plate square to the global X axis (shell elements, linear static)

The strip of shell_cantilever_edge, stood in the YZ plane so that its normal is the global X axis: code_aster's default reference axis (X) cannot orient such an element ('the plate element cannot be oriented'); the runner now gives each face a reference axis that suits it. 1000 N along X on the free edge: the same numbers as the strip in the XY plane.

Geometry	A STEP file (yz.step)
Analysis	Linear static, shell elements (thin plates)
Shell thickness	10 mm
Material	Steel: E = 210 GPa, Poisson's ratio 0.3, density 7850 kg/m3
Mesh	element size 25 mm, linear elements
Support	Fixed support on the faces or edges in the plane Z = 0 mm
Load	Force on the faces or edges in the plane Z = 1000 mm (FX (N) = -1000)

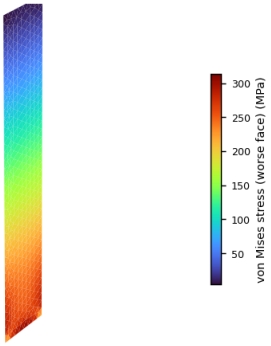


Geometry, supports and loads



802 surface triangles, 450 surface nodes

Mesh



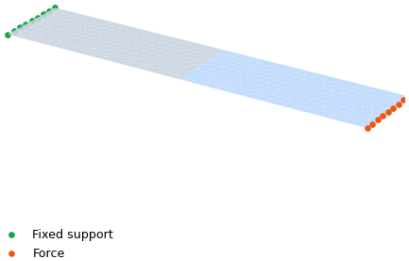
Main result

Quantity	Reference	cFEA	Difference	Allowed	Result
Maximum displacement Reference: P L^3 / (3 E I), I = 200 x 10^3 / 12	95.24 mm	93.32 mm	-2.01 %	4 %	**Pass**
Reaction force, X Reference: reaction = applied load	1,000 N	1,000 N	0 N	+/- 0.01 N	**Pass**
Maximum von Mises stress Reference: 6 P L / (b t^2) = 300 MPa at the root	300 MPa	313.4 MPa	+4.46 %	10 %	**Pass**

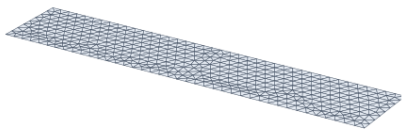
Plate cantilever with two thicknesses (shell elements, linear static)

Steel strip 1000 x 200 mm in two faces: 10 mm thick from the clamped edge to x = 500, 6 mm thick from there to the free edge, 1000 N along the free edge. Euler-Bernoulli beam with a step in EI (the shell has no shear deformation, and the two mid-surfaces coincide).

Geometry	A STEP file (stepped.step)
Analysis	Linear static, shell elements (thin plates)
Shell thickness	10 mm; face 2: 6 mm
Material	Steel: E = 210 GPa, Poisson's ratio 0.3, density 7850 kg/m3
Mesh	element size 25 mm, linear elements
Support	Fixed support on the faces or edges in the plane X = 0 mm
Load	Force on the faces or edges in the plane X = 1000 mm (FZ (N) = -1000)

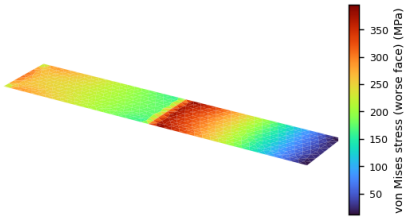


Geometry, supports and loads



804 surface triangles, 451 surface nodes

Mesh



Main result

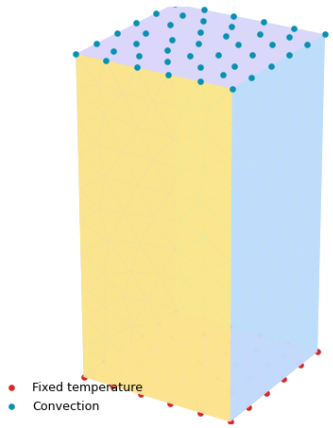
Quantity	Reference	cFEA	Difference	Allowed	Result
Maximum displacement Reference: P [(L^3 - (L-a)^3) / (3 E I1) + (L-a)^3 / (3 E I2)], I = 200 t^3 / 12 (t = 10 then 6), a = 500	138.4 mm	135.8 mm	-1.93 %	5 %	**Pass**
Reaction force, Z Reference: reaction = applied load	1,000 N	1,000 N	0 N	+/- 0.01 N	**Pass**
Mass Reference: rho x (0.5 x 0.2 x 0.010 + 0.5 x 0.2 x 0.006)	12.56 kg	12.56 kg	+0.00 %	0.01 %	**Pass**

Steady-state heat conduction

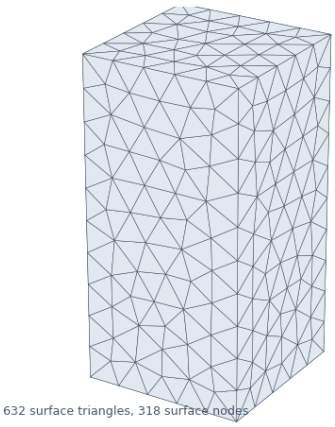
Block, fixed temperature and convection (steady-state thermal)

1D conduction through 200 mm of steel (k = 50) from 100 C to a surface cooled by h = 500 W/m2K at 20 C.

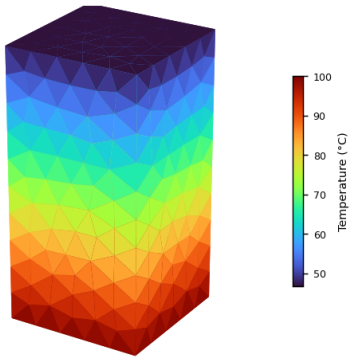
Geometry	Box 100 x 100 x 200 mm
Analysis	Steady-state thermal, solid elements
Material	Steel: density 7850 kg/m3, conductivity 50 W/(m.K)
Mesh	element size 20 mm, quadratic elements
Support	Fixed temperature on the faces or edges in the plane Z = 0 mm (T (degC) = 100)
Load	Convection on the faces or edges in the plane Z = 200 mm (h (W/m2.K) = 500, ambient temperature (degC) = 20)



Geometry, supports and loads



Mesh



Main result

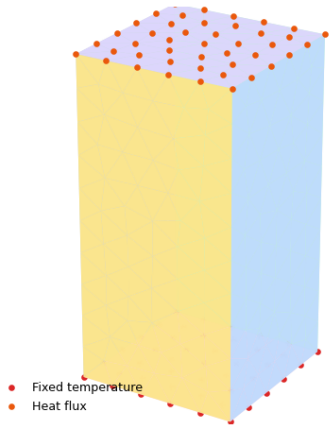
Quantity	Reference	cFEA	Difference	Allowed	Result
Minimum temperature Reference: T_inf + q/h, q = (T0 - T_inf)/(L/k + 1/h)	46.67 degC	46.67 degC	0.00 %	0.01 %	**Pass**
Maximum heat flux Reference: q = 80 / (0.004 + 0.002)	13,333 W/m2	13,333 W/m2	+0.00 %	0.01 %	**Pass**

Block, heat flux (steady-state thermal, geometry in metres)

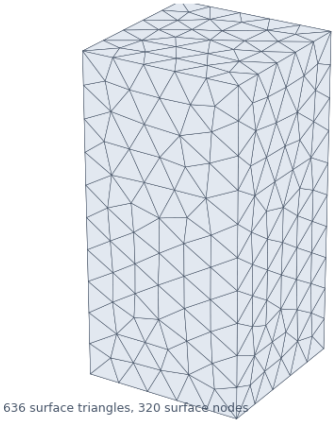
1000 W/m2 into one end of a 0.2 m block (k = 50), the other end at 20 C. Also checks the unit conversion from metres.

Geometry	Box 0.1 x 0.1 x 0.2 m

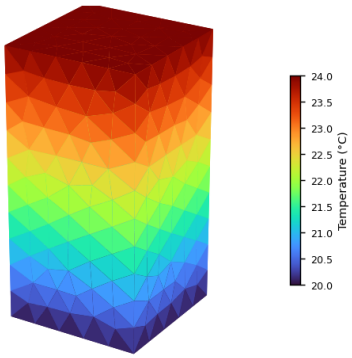
Analysis	Steady-state thermal, solid elements
Material	Steel: conductivity 50 W/(m.K)
Mesh	element size 0.02 m, quadratic elements
Support	Fixed temperature on the faces or edges in the plane Z = 0 m (T (degC) = 20)
Load	Heat flux on the faces or edges in the plane Z = 0.2 m (q (W/m2) = 1000)



Geometry, supports and loads



Mesh



Main result

Quantity	Reference	cFEA	Difference	Allowed	Result
Maximum temperature Reference: T0 + q L / k	24 degC	24 degC	+0.00 %	0.01 %	**Pass**
Maximum heat flux Reference: flux in = flux through	1,000 W/m2	1,000 W/m2	+0.00 %	0.01 %	**Pass**

Differences are shown as (cFEA - reference) / reference. Reference values come from the formula named under each quantity. Report generated 2026-11-18 from the verification suite of cFEA.