NAFEMS Benchmark Problem 02 Solution

Matt Watkins

Name of challenger	Matt Watkins
Roark (Collapse Load in kN/m^2)	211
Steel Designer’s Manual (Collapse Load in kN/m^2)	103
Name/Version of FE System used	StressCheck 10.1
Name of element used (and integration scheme etc)	Hex elements with 8th order shape functions in-plane and 2nd order shape functions thru-thickness. Fully integrated.
BM1 (Collapse Load in kN/m^2)	
BM2 (Collapse Load in Force/Area)	0.0188
BM3 (Collapse Load in kN/m^2)	
Yield Line Approximation (Collapse Load in kN/m^2)	
Yield Line Approximation (value of d[m] used)	
Challenge Problem – Load at first yield (Load in kN/m^2)	Approx. 120
Challenge Problem – Load at collapse (Collapse Load in kN/m^2)	Approx. 230

Blank cells above indicate that the corresponding approaches were not attempted, since they were not required for answering the question.

1 Element Type

StressCheck has plate elements for linear analysis only, but supports plastic analysis (deformation and incremental theories of plasticity) with 3D elements. In order to solve the benchmark problem 3D elements were required. However, boundary conditions such as “simply supported” do not exist in 3D elasticity, therefore it was necessary to use a set of boundary conditions that approximated simple supports.

At simply supported boundaries the plate will tend to rotate as if the boundary was connected to a hinge. For a thin 3D domain this can be effectively represented with an antisymmetry constraint which prevents tangential displacement (including out-of-plane displacement) and allows normal displacement (in-plane displacement). The out-of-plane displacement restriction allows the boundary to react the distributed load while the in-plane displacements (expected to be linear about the neutral axis) represent rotation.

The second plasticity benchmark, from NAFEMS documentation, was used to check that antisymmetry constraints could be used to simulate a hinge. A linear solution to the given problem was computed first with plate elements using simple supports and then with solid hex elements using antisymmetry. Figures 1 and 2 show convergence of the displacement at the center of the plate for both cases (solution verification).

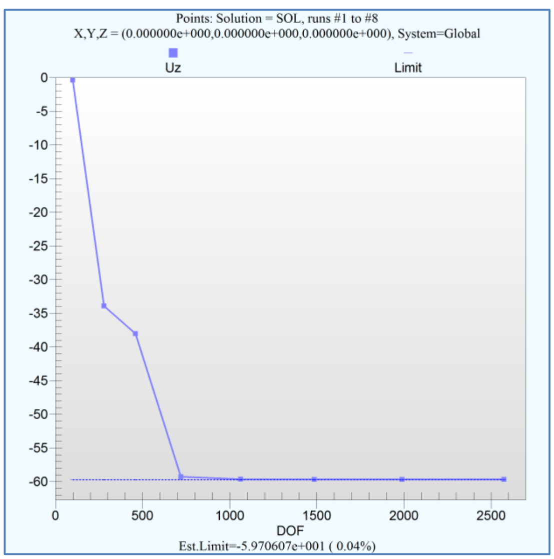


Figure 1: Convergence of displacement for plate elements with simple supports (verification)

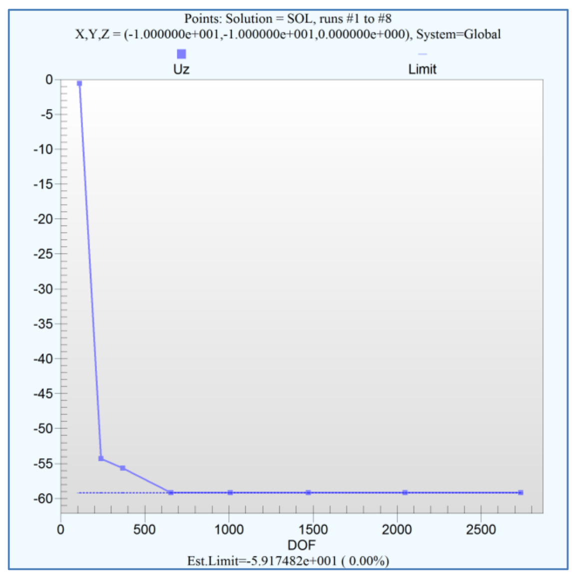


Figure 2: Convergence of displacement using 3D elements with antisymmetry constraints (verification)

The two modeling approaches can only be compared if the error of approximation is shown to be negligible for both approaches independently. Since the estimated error in displacement was less than 0.05% for both plate and 3D elements, the displacements can be compared: The displacement difference between the two approaches was less than 1%, showing that antisymmetry is an effective way to represent simple a hinge-like boundary condition in 3D elasticity.

A plasticity simulation was performed using 3D elements as a comparison with the published NAFEMS benchmark solution. The load-deflection curve is given in Figure 3.

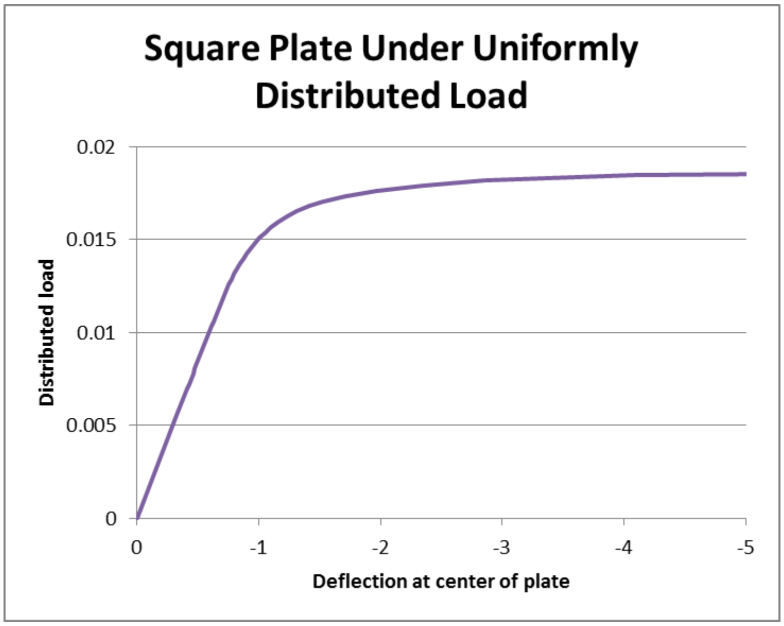


Figure 3: Comparison for NAFEMS benchmark using 3D elements

The load-deflection curve computed using 3D elements compares nicely with the reference solution published by NAFEMS with plate elements. The load limit was found to be approximately 0.01877.

It was therefore concluded that the analysis could proceed using antisymmetry constraints in place of simple supports.

2 Modeling Decisions

- Domain: 1/8 model.
- Constraints: Antisymmetry was used to simulate hinges. Symmetry was used to reduce the size of the domain in the x and y (in-plane) directions and antisymmetry was used to reduce the size of the domain in the z direction (out-of-plane).
- Loads: Distributed loading, half applied to the top surface and half applied to the bottom surface. This approximates a plate distributed load which is mathematically represented at the neutral surface of the plate.

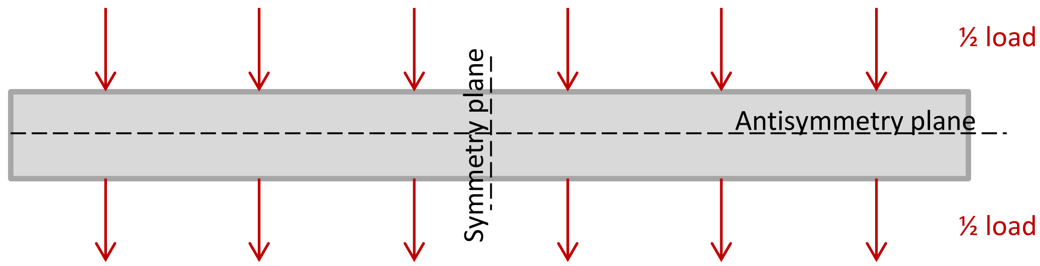


Figure 4: Loads and Constraints

- Mesh: 16 element hex mesh, graded toward the boundaries (Figure 5).

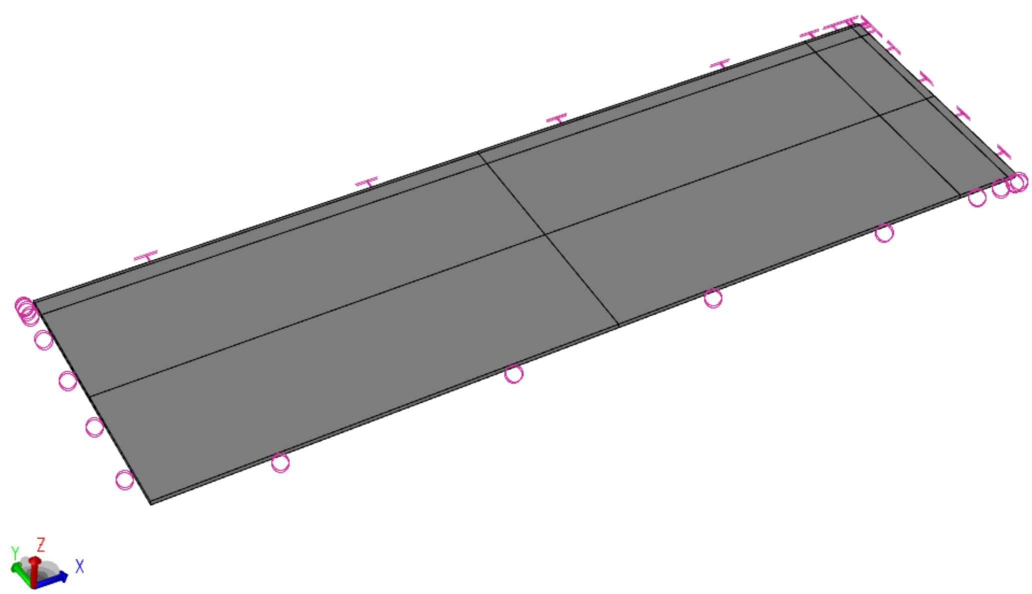


Figure 5: Finite element mesh for representing the problem.
'T' shapes represent antisymmetry constraints and circles represent symmetry constraints.

- Material: elastic perfectly-plastic (E=200 GPa, v=0.295, σ_Y=275 MPa)

3 Verification

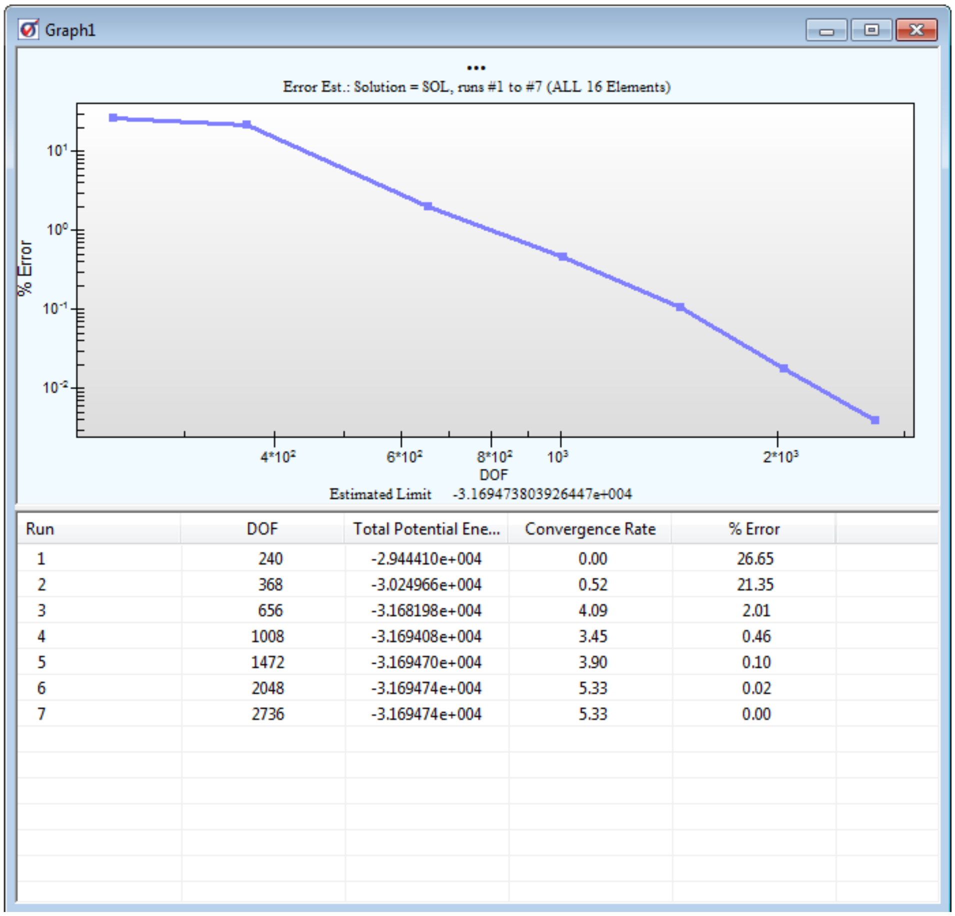
The problem statement asks for evidence of verification, which encompasses code verification and solution verification. Solution verification is an assessment of the error of approximation of the finite element solution. The error of approximation is the difference between the computed solution and the exact solution of the problem (whether the exact solution is known, or not).

Two comments:

- The second set of hints states that "the number of theoretically exact solutions for linear-elastic problems is fairly small". I would clarify that the number of *analytically-obtained* exact solutions is fairly small. Every solvable problem has a theoretically exact solution – the approximate FEA solution will converge to it as DOF go to infinity.
- The only type of verification that is mentioned in the hints is code verification, which is much less important than solution verification for FEA end-users. Code verification is the responsibility of the FEA software company, that is, ensuring that the code can compute what it claims to compute without bugs. Solution verification must be performed for every single problem that is solved. A software program may be perfectly capable of reliably solving problems, but the quality of the solution depends on the number of degrees of freedom used[1]. Most FEA software can do this with mesh refinement -- verification evidence is therefore objective proof that the mesh is dense enough for the given problem (a "reasonable" mesh density is rarely, if ever, known beforehand).

The typical procedure for producing verification evidence for nonlinear problems is to ensure that the corresponding linear solution has a very small error of approximation. Figure 6 shows convergence information for the energy norm of the linear solution as DOF increase. This is an estimate of global error for the entire mesh. As shown, with 2736 DOF the global error estimate is less than 0.005% which is more than sufficient.

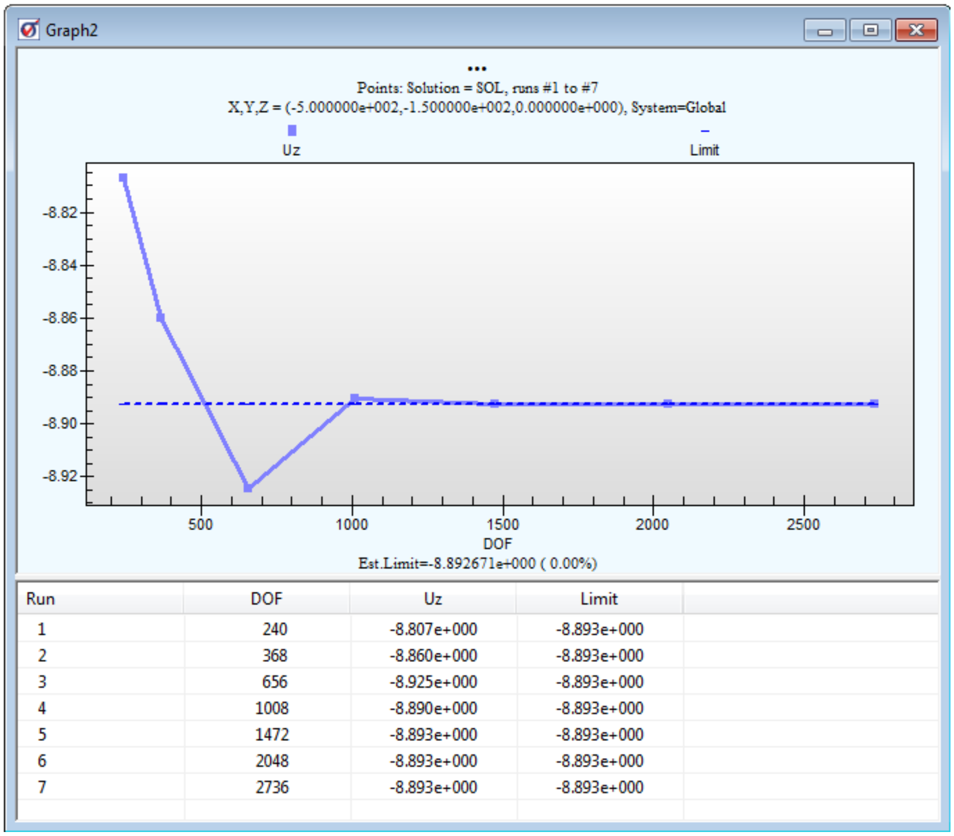
[1] And the mapping, which must well-represent the domain.



Run	DOF	Total Potential Ene...	Convergence Rate	% Error
1	240	-2.944410e+004	0.00	26.65
2	368	-3.024966e+004	0.52	21.35
3	656	-3.168198e+004	4.09	2.01
4	1008	-3.169408e+004	3.45	0.46
5	1472	-3.169470e+004	3.90	0.10
6	2048	-3.169474e+004	5.33	0.02
7	2736	-3.169474e+004	5.33	0.00

Figure 6: Convergence of energy norm

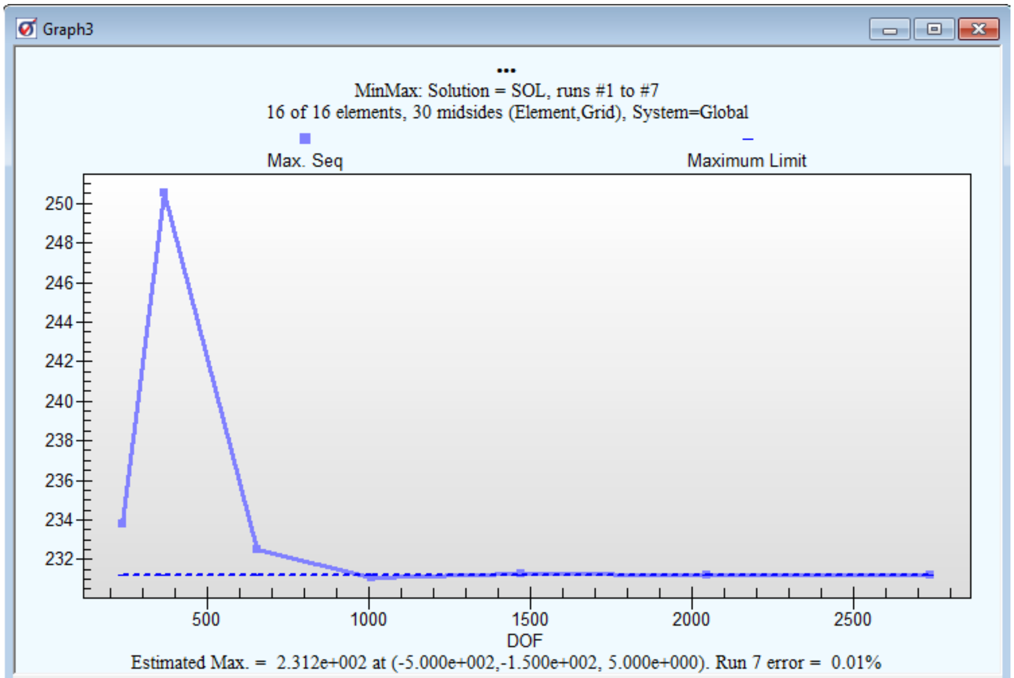
Figure 7 shows convergence of the displacement at the midplane at the center of the plate. Again, the estimated error is less than 0.005%, indicating that the solution has converged to the exact solution.



Run	DOF	Uz	Limit
1	240	-8.807e+000	-8.893e+000
2	368	-8.860e+000	-8.893e+000
3	656	-8.925e+000	-8.893e+000
4	1008	-8.890e+000	-8.893e+000
5	1472	-8.893e+000	-8.893e+000
6	2048	-8.893e+000	-8.893e+000
7	2736	-8.893e+000	-8.893e+000

Figure 7: Convergence of displacement

In general it is easier to converge on displacement than on stress, since the typical FEM implementation is the displacement formulation. Therefore convergence of each stress component was verified. Figure 8 shows convergence of the maximum von Mises stress. The estimated error for the final solution was 0.01%, indicating that the stresses of the exact solution were being well-approximated. Again, this very small error indicates that the mesh and polynomial order is more than sufficient to capture the solution, suggesting that the corresponding nonlinear solution will also have low error.



Run	DOF	Max. Seq	Maximum Limit	X	Y
1	240	2.338e+002	2.312e+002	-5.000e+002	-1.500e+002
2	368	2.505e+002	2.312e+002	-5.000e+002	-1.500e+002
3	656	2.325e+002	2.312e+002	-5.000e+002	-1.500e+002
4	1008	2.311e+002	2.312e+002	-4.839e+002	-1.500e+002
5	1472	2.313e+002	2.312e+002	-5.000e+002	-1.500e+002
6	2048	2.312e+002	2.312e+002	-5.000e+002	-1.500e+002
7	2736	2.312e+002	2.312e+002	-5.000e+002	-1.500e+002

Figure 8: Convergence of von Mises stress

The solution with 2736 DOF corresponds to 8th order polynomial shape functions in-plane and 2nd order polynomial shape functions thru-thickness. This was the discretization used for all nonlinear solutions.

4 Results

A material nonlinear solution was computed, incrementally increasing the distributed load until the nonlinear solution could no longer converge. The resulting load-deflection curve is shown in Figure 9.

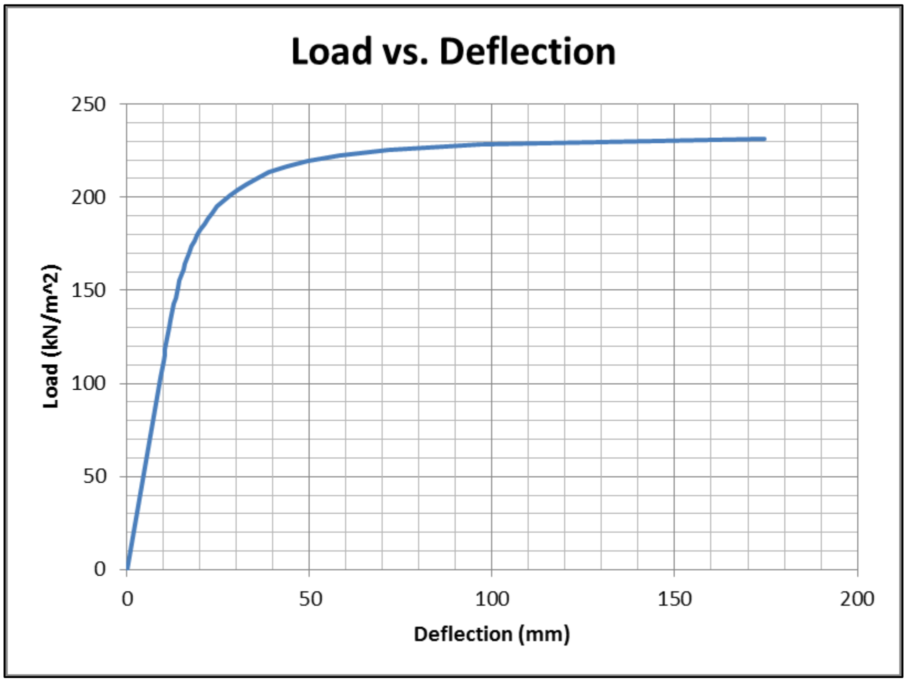
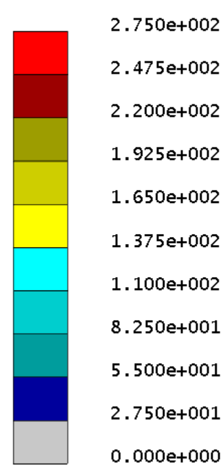


Figure 9: Load vs deflection for the benchmark problem

The following plots show the von Mises distribution through the plate as the load increases. The plot legend goes from 0 to 175 MPa. The lower-left corner of each plot is the center of the plate (at the point of double symmetry).

Also included is a plot of the von Mises stress through half of the thickness at the center of the plate. These show that the plastic zone grows toward the midplane from the top surface.

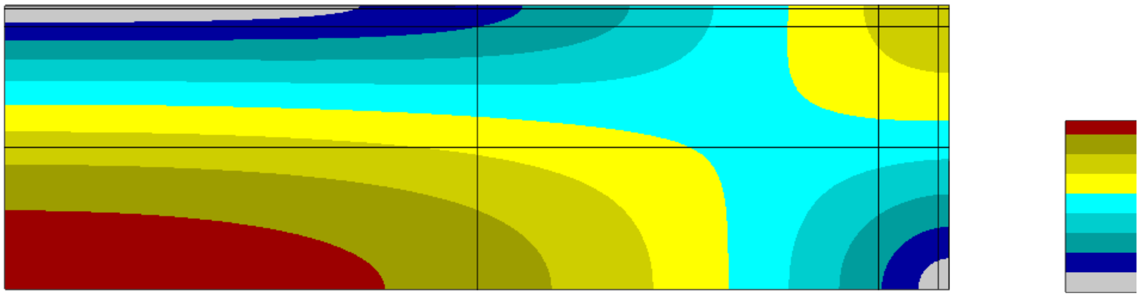


103 kN/m²

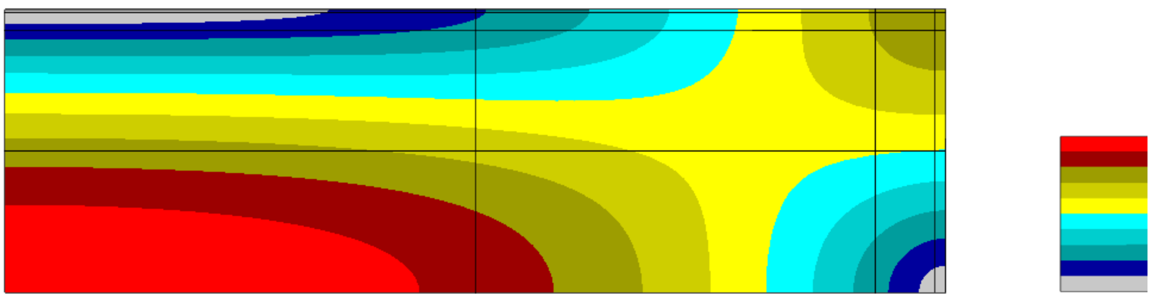
Top surface von Mises

Thru-thickness von Mises

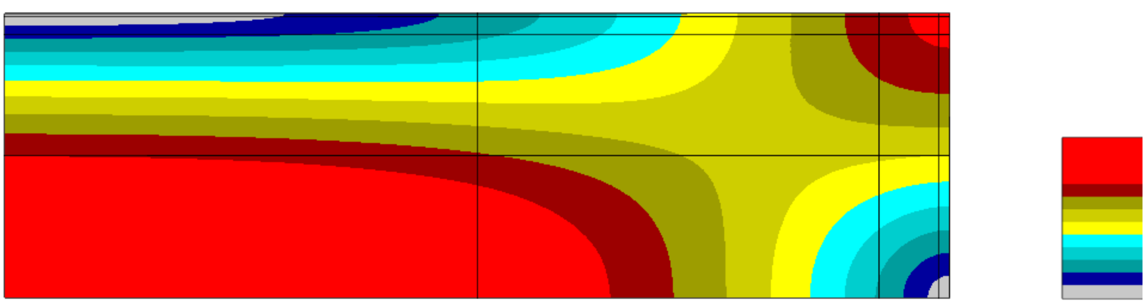
(half-thickness shown)



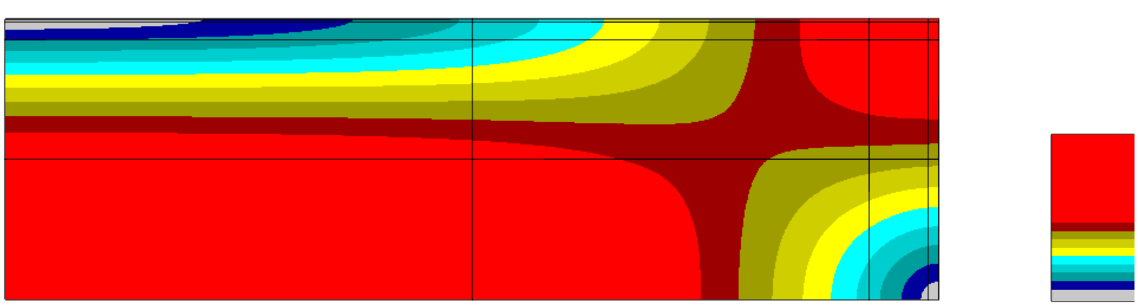
118 kN/m² (yielding begins)



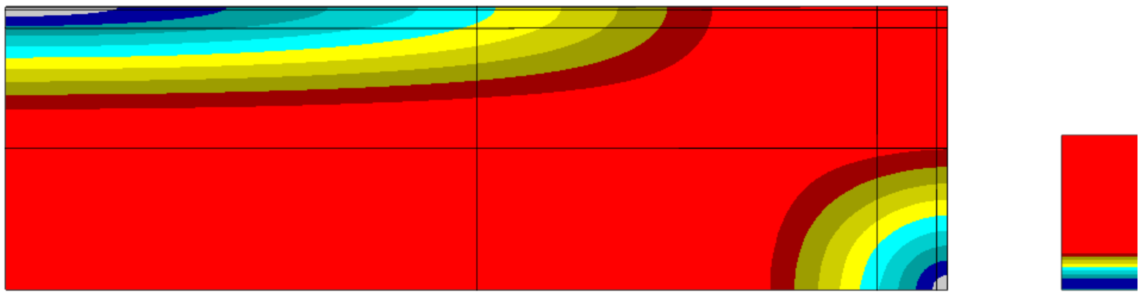
142 kN/m²



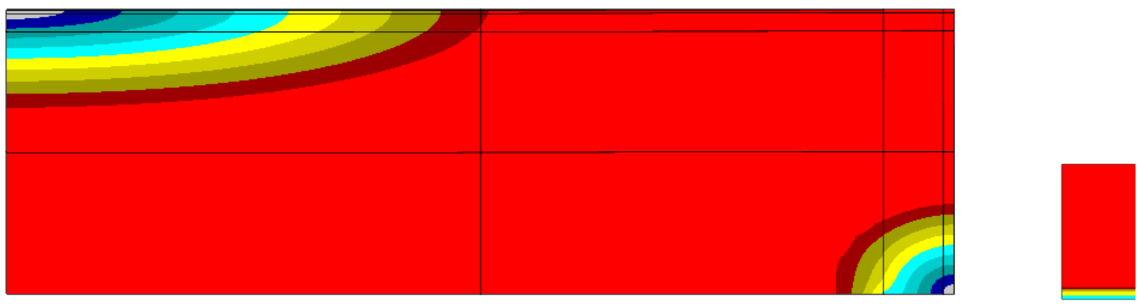
170kN/m²



195 kN/m²



219 kN/m²



231 kN/m² (collapsing)

The final collapse load is approximately 230 kN/m², which is close to the value given by Roark. Why the difference? The FEM solution presented is a higher-order model than the Roark solution which was likely obtained with an analytical approximation. As such, the Roark solution is expected to be conservative.

5 Recommendation

The value provided by Roark assumes that the edges cannot displace out-of-plane. In order for this to be an effective representation of reality the plate would have to be held in place by narrow slots all around the edges. The value provided by the Steel Designers' Manual allows the corners to lift, but is likely based on the first yield of the material in order to provide more conservatism. If the plate is not held in place by narrow slots then the smaller value from the Steel Designers' Manual can be used with the understanding that it is overly conservative.

6 Just for Fun: Nonlinear Elasticity Allowing Corners to Lift

As a complement, an analysis was performed using general nonlinear elasticity (large strain, large displacement) to allow membrane stiffening effects. The edges and corners of the plate were allowed to lift and slide with the use of distributed nonlinear springs which only take compressive loads.

A note about realism: Clearly, the material cannot take increasing load forever. At some point the elastic-perfectly-plastic curve ceases to be a reasonable representation of the true material behavior, and of course the material cannot be expected to carry more than its ultimate load. I do not know the ultimate stress or strain point for this particular steel, but, for reference, the maximum first principal strain for the entire body is plotted in Figure 11. Considering the ridiculously large corresponding strains at the higher load values, it is likely the material would tear and fail much before the load values shown. Just because a model will solve and produce a reasonable-looking solution doesn't mean it has anything to do with reality.

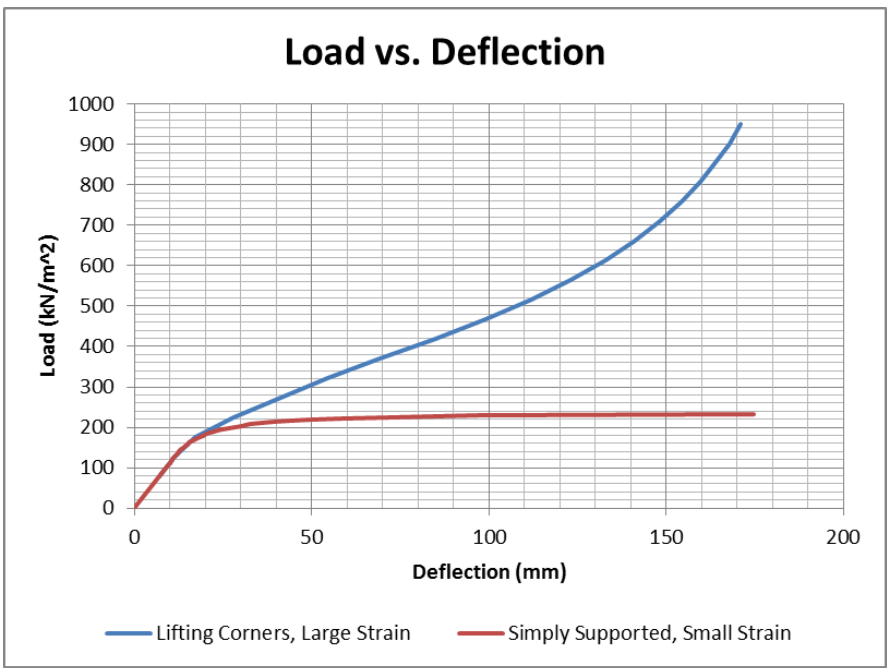


Figure 10: Comparison of load vs. deflection for lifting corners and large strain elasticity and the previous results

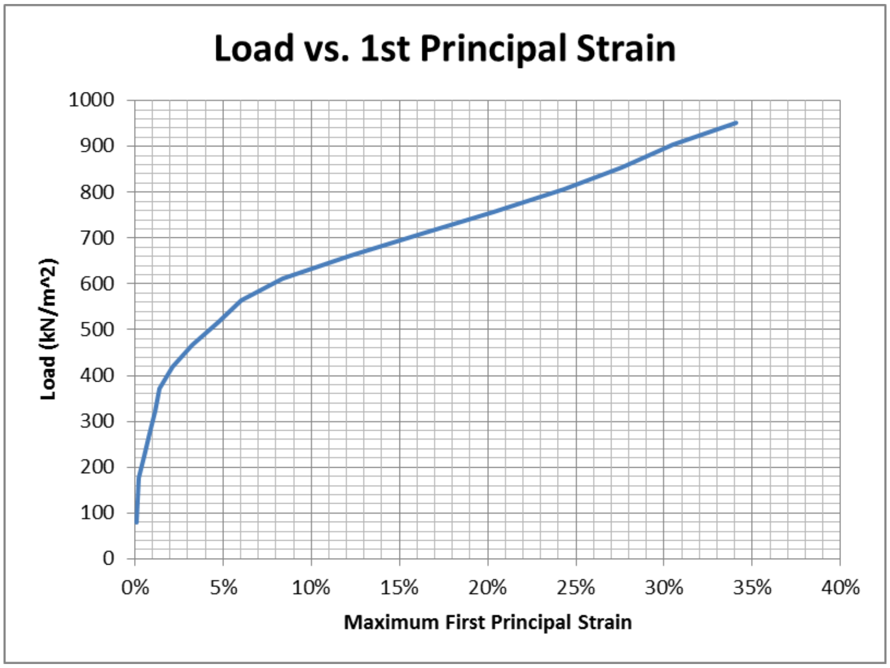


Figure 11: Load vs First Principal Strain

It's worth mentioning that the upper and lower surfaces are completely plastified by 320 kN/m^2, corresponding to 1% maximum first principal strain (the midplane is not yet completedly plastified). A more reasonable comparison would probably be given by Figure 12.

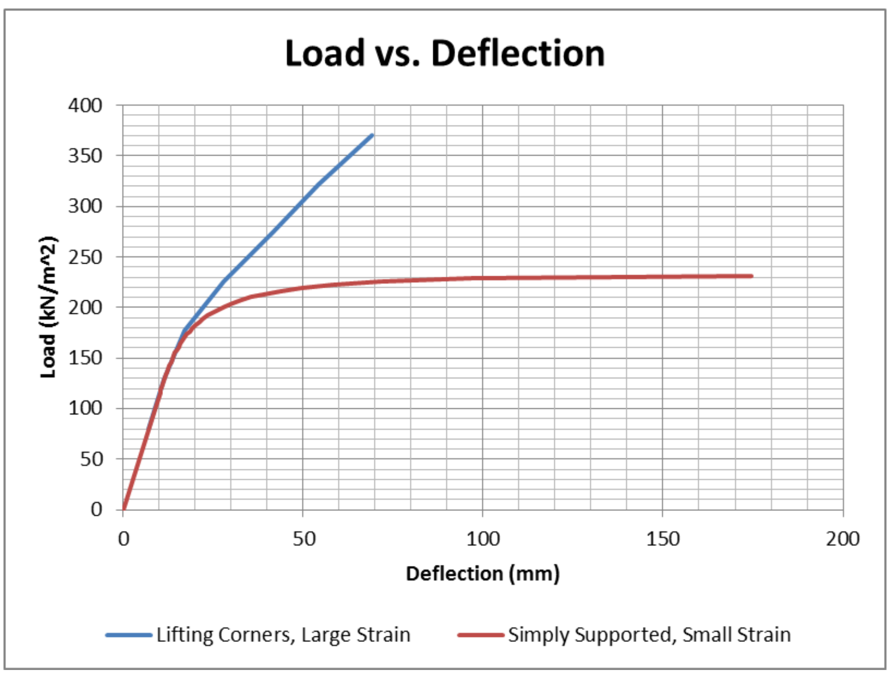


Figure 12: Comparison of load vs. deflection for lifting corners and large strain elasticity and the previous results