NAFEMS Benchmark Testing Versus Abaqus Models, A Comparison

Submitted by:

Adam Michalak

FEGN 526 – Static & Dynamic Applications in FEA

December 16, 2020

Executive Summary

NAFEM's benchmark study utilized 2D elements to analyze a part in a plane stress situation. This study was used to compare 3D shell elements to. Overall, the 3D shell elements performed extremely well with a high level of accuracy and precision. The 3D shell element's meshes converged quickly and at a low element number. This report displays the possibility that 3D shell elements could be used to accurately perform FEA on 2D/3D shell parts such as sheet metal.

Introduction

The National Agency for Finite Element Methods and Standards, which is commonly referred to as NAFEM, is known to set the standard across the industry for Finite Element Analysis. In this comparison 3D shell elements have been used to study the modeling accuracy when compared to 2D solid elements in a plane stress application. By comparing multiple element types from the 2D solid and 3D shell models one will be able to understand the practicality of modeling with 3D shell elements. To verify and validate this study three different element types will be studied including: CPS3, CPS4, and CPS8R. The 2D solid elements will be compared to S3, S4, and S8R, 3D shell elements. The 2D solid element models have already been modeled and analyzed by NAFEM as well as a real-world test which results have been compared to.

The purpose of this project is to perform a comparison between various element types for 3D shell models and to perform a benchmark comparison with 2D solid models and to an actual test performed by NAFEM.

Methods

For this benchmark comparison Dassault's SIMULIA Abaqus 2020 software version was used to model, mesh, and analyze the 3D shell model that utilized various element types. The part that was analyzed was modeled in reference to Dassault's SIMULIA Abaqus documentation. Below the 2D geometry of the model is illustrated in Figure 1. The only difference between the 2D model in Figure 1 and the 3D shell part, was that the 3D shell part was created as a shell with a tenth of a meter thickness.

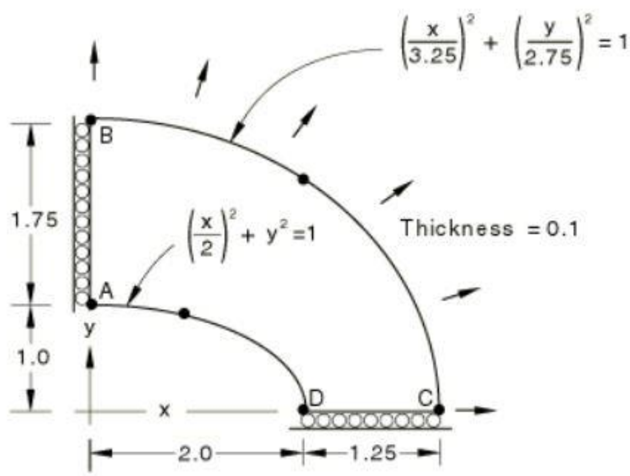

Figure 1. 2D geometry of the model which was used for a benchmark comparison. Pinned-roller style boundary condition is also displayed at points A, B, C, and D.

Displayed in Figure 2 below is the 3D part that was modeled for the purpose of performing this benchmark comparison. The file name "QuarterCirclePart.CAE" was used to store this file.

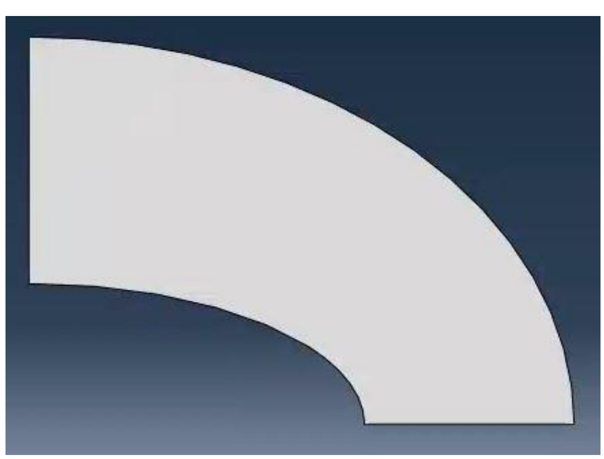

Figure 2. "Quarter Circle Part" modeled in SIMULIA Abaqus.

Referring to Figure 1, Boundary conditions were applied in the U_X and U_Z directions for the line connecting points A and B on the Quarter Circle Part. Boundary conditions were also applied in the U_Y and U_z directions for the line connecting points C and D. For all points, A, B, C, and D, the U_Z direction was constrained due to this model representing a 2D situation. Therefore, U_Z was constrained to create an accurate analysis of this model since 3D shell elements were used.

In Figure 4, the shell load is illustrated along with the boundary conditions that were applied to the model. The shell edge load was applied at a force of 10 MPa. The material that was modeled in the Quarter Circle Part was A-36 Steel which has a Young's Modulus of 210 Gpa, 0.3 Poisson's Ratio, and a density of 7800 kg/m^3.

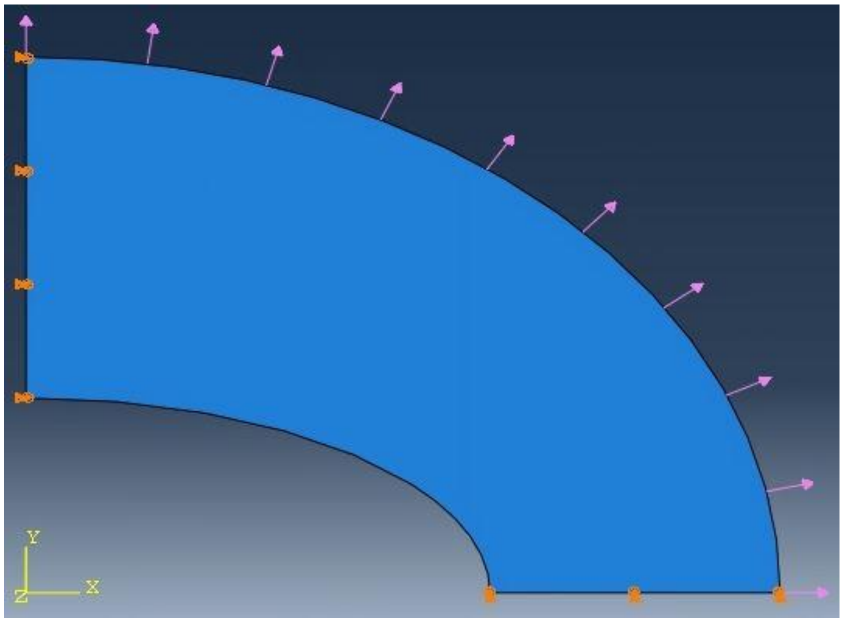

Figure 4. Shell Edge Load applied to Quarter Circle Part with boundary conditions applied.

For the Finite Element Analysis of the model SIMULIA Abaqus was also used. The mesh was generated by seeding the four edges of the part with a specific number of elements. When the mesh would be refined for convergence the number of elements seeded on the edge would increase by 10. The initial mesh for each element types used for comparison in this report was seeded with 10 elements per edge. This amount would increase by 10 elements per edge until the mesh was refined. To perform a reasonable comparison between NAFEM's study and the results generated in this report element types of similar nature were used. With the primary difference between 2D element types versus 3D shell elements. The element types used from the NAFEM's study for comparison were the following: CPS3, CPS4, and CPS8R. The 3D shell elements generated by the author of this report were the following: S3, S4, and S8R. Mesh convergence was considered and studied. For mesh convergence, the mesh would be applied to the Quarter Circle Part as described above and the maximum von Mises stress used as a data point. In addition, the node location would be used to ensure that the max stress was not changing locations, thus creating an inaccurate model. The maximum von Mises stress was found for every mesh type at point D as seen in Figure 1 above. A mesh was considered converged when the maximum stress did not differ by greater than 5%. An example of the type of mesh used can be found in Figure 5 below.

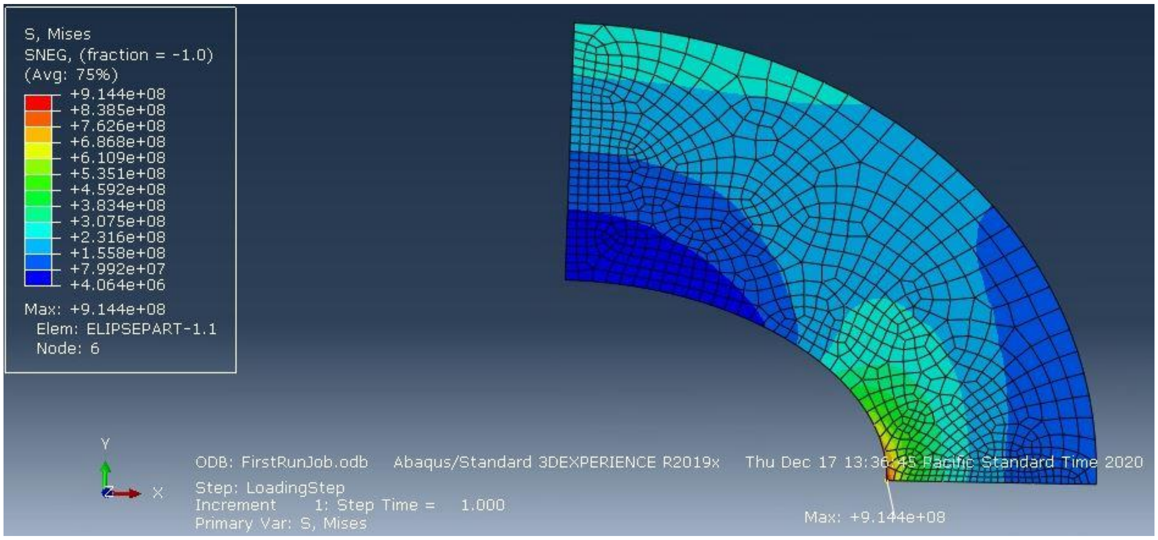

Figure 5. A highly refined mesh that highlights the edge's seeded with a specific number of elements to study mesh convergence.

Results

The comparison study of NAFEM's 2D elements (CPS3, CPS4, and CPS8R) and this report's 3D shell elements (S3, S4, and S8R) will be analyzed along with a mesh convergence study.

The first element type to be reported on will be the 3D shell element type 'S3'. The results for this element's mesh convergence can be found below in Table 1. This element will be compared to element type CPS3 at the end of the results section.

Table 1. Mesh convergence study for the S3 shell element.

	Number of Elements	Peak von Mises Stress (Pa)
Mesh 1	156	5.98E+08
Mesh 2	584	8.94E+08
Mesh 3	920	9.11E+08

The percent difference between S3 Mesh 2 and S3 Mesh 3 was +1.88%. The image of the S3 mesh with peak von Mises stress displayed in Pascals is below in Figure 6.

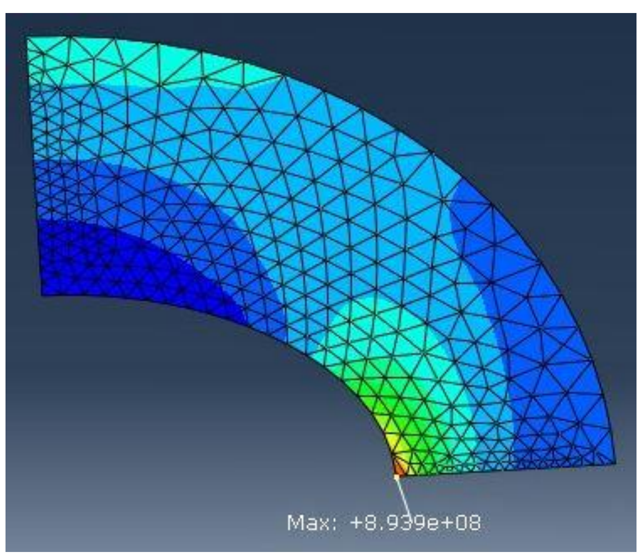

Figure 6. S3 Element Type converged mesh with 20 elements across each edge. A peak von Mises stress is displayed at point D.

The next element type to be discussed will be the S4 element type. The mesh convergence results for this element type can be seen below in Table 2. The S4 element type will be compared to the CPS4 element type from NAFEM's study.

Table 2. Mesh convergence study for the S4 shell element.

	Number of Elements	Peak von Mises Stress (Pa)
Mesh 1	153	8.20E+08
Mesh 2	348	8.94E+08
Mesh 3	864	9.14E+08

The percent difference between S4 Mesh 2 and S4 Mesh 3 was +2.43%. The image of the S4 mesh with peak von Mises stress displayed in Pascals is below in Figure 7.

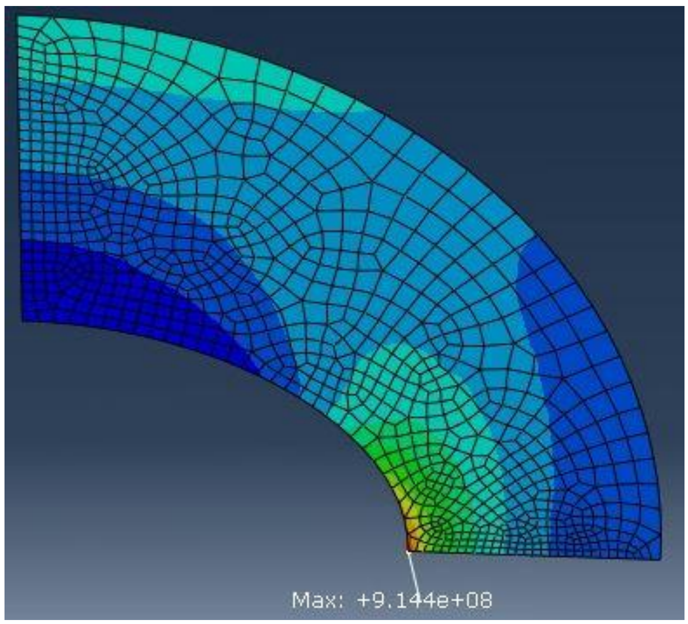

Figure 7. S4 Element Type converged mesh with 20 elements across each edge. A peak von Mises stress is displayed at point D.

The next element type to be discussed will be the S8R element type. The mesh convergence results for this element type can be seen below in Table 3. The S8R element type will be compared to the CPS4 element type from NAFEM's study.

Table 3. Mesh convergence study for the S8R shell element.

	Number of Elements	Peak von Mises Stress (Pa)
Mesh 1	23	9.66E+08
Mesh 2	82	9.55E+08
Mesh 3	346	9.38E+08

The percent difference between S8R Mesh 2 and S8R Mesh 3 was -1.15%. The image of the S8R mesh with peak von Mises stress displayed in Pascals is below in Figure 8. It should also be noted that the number of elements for S8R converged at a significantly lower number than the other elements.

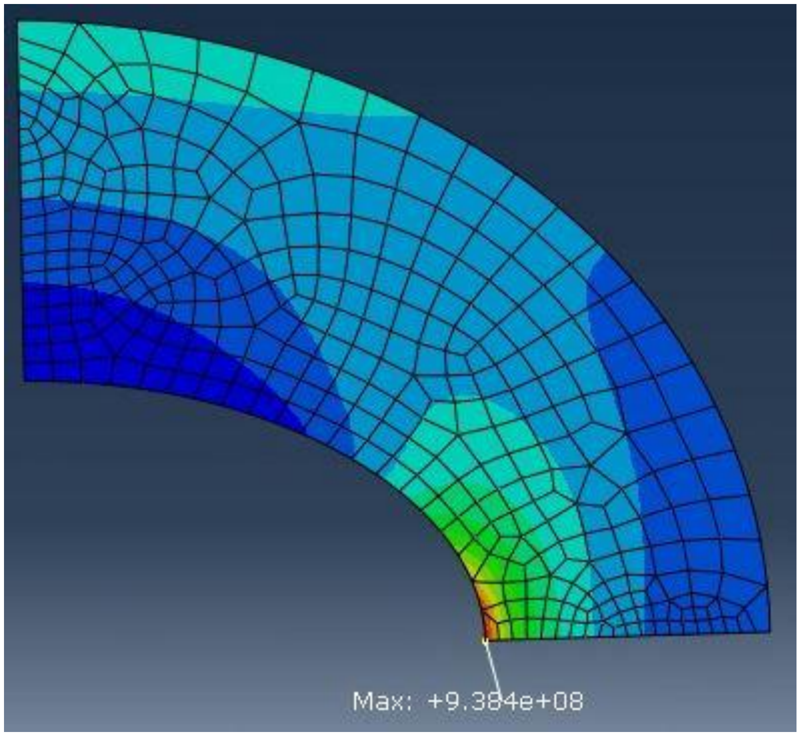

Figure 8. S8R Element Type converged mesh with 20 elements across each edge. A peak von Mises stress is displayed at point D.

Below are the results of the above-mentioned 3D shell elements compared to NAFEM's benchmark test. The peak edge stress in NAFEM's Benchmark study was 92.7 MPa at point D. The Fine Meshes have been compared from CPS3, CPS4, and CPS8's percent difference to S3, S4, and S8's percent difference. To be clear, this is the percent difference from these elements compared to NAFEM's 92.7 MPa which they achieved in October 1990. The results can be seen below in Table 4(a) and Table 4(b).

Table 4(a). 2D Element's percent difference compared to NAFEM's benchmark.

2D Elements

Element Type	Percent Difference
CPS3	-23%
CPS4	-9%
CPS8R	0.12%

Table 4(b). Mesh convergence study for the S8R shell element.

3D Shell Elements

Element Type	Percent Difference
S3	1.88%
S4	2.43%
S8R	-1.15%

Discussion

The purpose of this benchmark study was to compare 3D shell elements to NAFEM's benchmark study which utilized 2D elements. The results displayed a high level of accuracy compared to NAFEM's 1990 study. Each of the shell elements converged at a low element number and the percent difference to the benchmark study was remarkably low. This means that modeling this Quarter Circle Part in 3D as a shell and utilizing Abaqus to mesh and analyze the part under load shows results with a high level of accuracy and precision. This means that the 3D shell elements analyzed in this report could be appropriate in certain situations to achieve an accurate and precise analysis. The S8R element type was specifically interesting as the mesh converged quickly, at a low element count, and was highly accurate and precise compared to NAFEM's benchmark study. The verification and validation of these elements can already be spoken to as this was already a direct comparison to NAFEM's study which performed verification and validation with the 2D elements mentioned in this report.

References

1. National Agency for Finite Element Methods and Standards (U.K.): Test LE1 from NAFEMS publication TNSB, Rev. 3, "The Standard NAFEMS Benchmarks," October 1990.

2. Smith, M 2009, *ABAQUS/Standard User's Manual, Version 6.9*. Dassault Systèmes Simulia Corp, Providence

Credits

The authors of this report would like to thank Dr. S. Geer and Dr. A. Petrella of Colorado School of Mines for laying the foundation for this comparison study.