CE422 Project 2

Author
Jared Jacobowitz

30 November 2021
CE422 - Professor Hapij

Contents

List of Figures

List of Tables

Problem Description

A 2D Bernoulli-Euler beam under external loading and boundary conditions of the student's choice was to be analyzed using commercial FEA software. Three mesh resolutions of 1x10, 2x20, and 4x40 using isoparametric bilinear quadrilateral elements were to be compared. Using FEA software, the beam was to be evaluated for stress, displacement, and reaction forces. The results were to be verified by hand calculations.

Modeling Considerations

The rectangular 2D Bernoulli-Euler beam modeled in this report is shown in Figure 1. The beam was modeled as $40m$ long, $4m$ tall, and $1m$ thick (not shown in the Figure). The beam was supported under simply supported conditions where the left end, at point A, was a pinned support, restricting the end from translating in the x or y directions, and the right end, at point B, was a roller support, only restricting the end from translating in the y direction. The beam was modeled as steel, which has a Young's modulus of $200 \times 10^9 Pa$ and Poisson ratio of 0.25.[1] The beam was loaded by a single $10kN$ downward point load on the top surface located halfway between the two ends (i.e. $20m$ from either end).

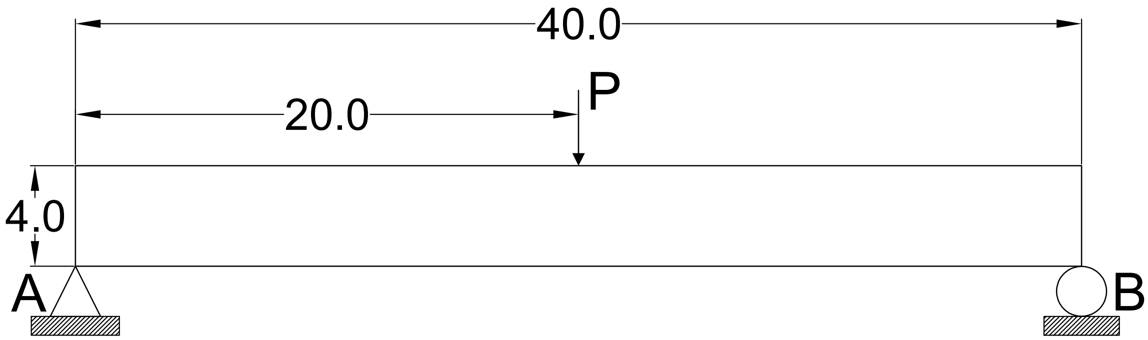

Figure 1: Rectangular 2D Bernoulli-Euler beam analyzed in this report. Not shown: $1m$ depth into the page.

Hand Calculations

Hand calculations were performed for the simply supported beam to verify the simulation results later in the report. The detailed hand calculations can be found in the Appendix under *Detailed Hand Calculations*. The simply supported beam is well understoo, and there exists equations for calculating the displacement (Equation (3)), reactions (Equaton 1), and stress (Equation 5) in the beam. The results are tabulated below in Table 1.

What	Value
Reaction Forces (R_A and R_B)	$5kN$ upward
Maximum Displacement (y_{max})	$0.125 \times 10^{-4}m$
Max Stress (σ_{max})	$37500Pa$

Table 1: Hand calculations results to be used as verification for the simulations.

Simulations

General Setup

The simulation was performed using Ansys APDL finite element software (Version: 2021 R1 Build 21.1). The SHELL181 element with default settings was chosen to mesh the beam. SHELL181 is a 4-node element with six degrees of freedom for each node, corresponding to three degrees of nodal translation and three degrees of nodal rotation.[2] A structural, linear, elastic, isotropic material was chosen, and the Young's modulus and Poisson's ratio for steel were input.

The beam was modeled as an area 40 units long, corresponding to $40m$ long, and 4 units tall, corresponding to $4m$ tall. The area was meshed with the shell elements with a layup thickness of 1 unit, corresponding to the $1m$ depth of the beam. The mesh size depended on the resolution being used; the mesh size is mentioned under the later sections. Because this is a 2D analysis, all nodes were restricted from displacing in the z direction or rotating out of plane in the x or y directions. Additionally, the bottom left node was restricted from translating in the x or y directions and the bottom right node was restricted from translating in the y direction, modeling a pinned and roller support, respectively. The appropriate loads were applied in the $-y$ direction at the node on the top surface which was equidistant from the left and right ends. All the settings used are summarized in the *APDL Inputs and Outputs* section.

The simulation was run using the built-in default structural solver and settings in Ansys APDL.

1x10

Results

The 1x10 resolution means the beam was meshed with 1 element in height and 10 elements along its length. Because the beam was modeled as 40 units long and 4 units tall, the 1x10 beam was meshed with elements that had edge lengths of 4 units (i.e. $4m$). This produced the mesh shown in Figure 2.

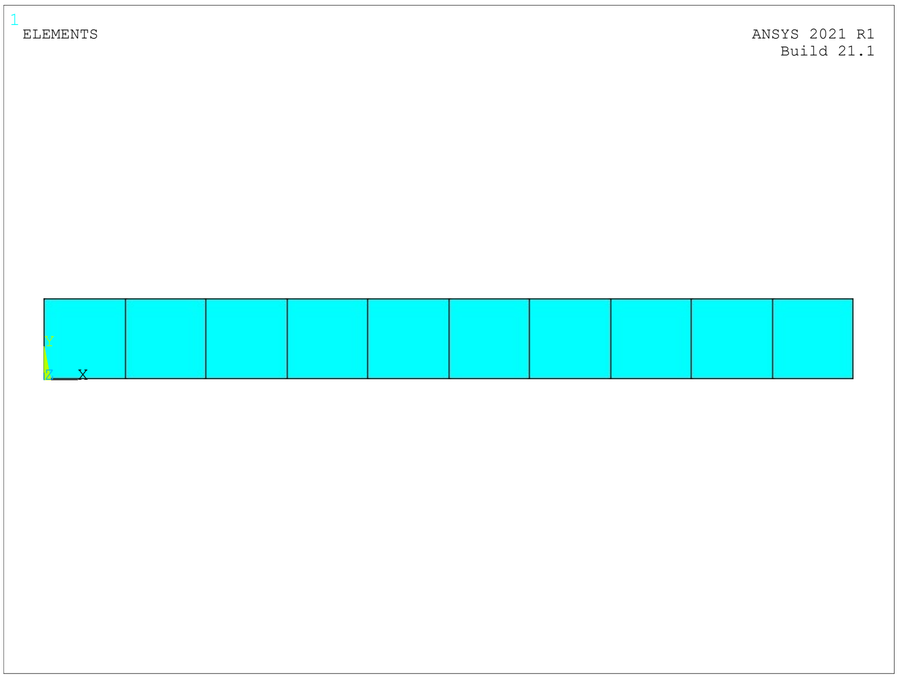

Figure 2: Mesh of the 1x10 beam.

Once simulated, the beam deforms as shown in Figure 3. The x stress and y displacement of this beam can be seen in Figures 4 and 5, respectively.

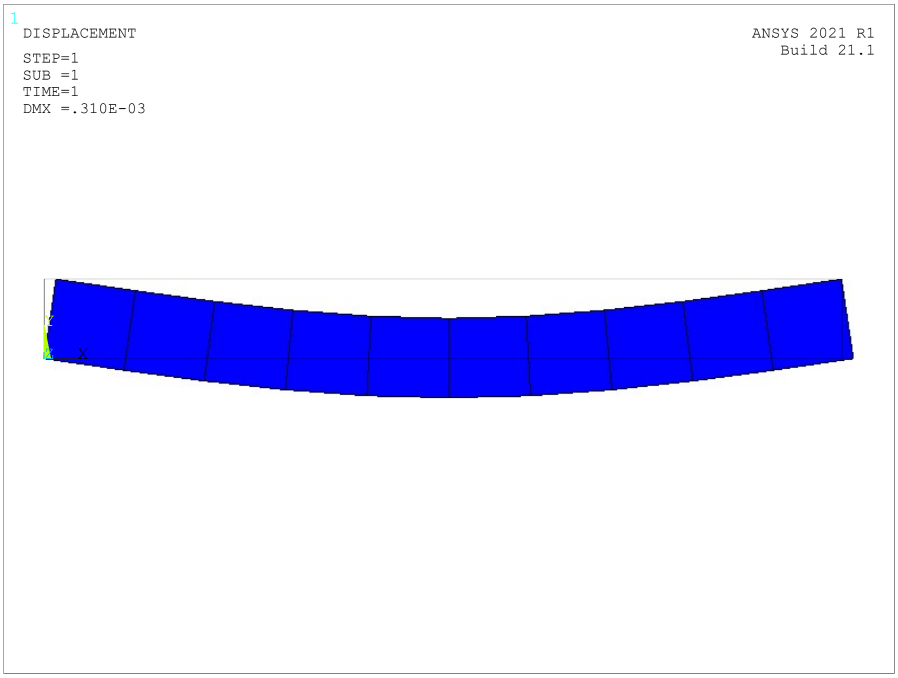

Figure 3: Deformed 1x10 beam.

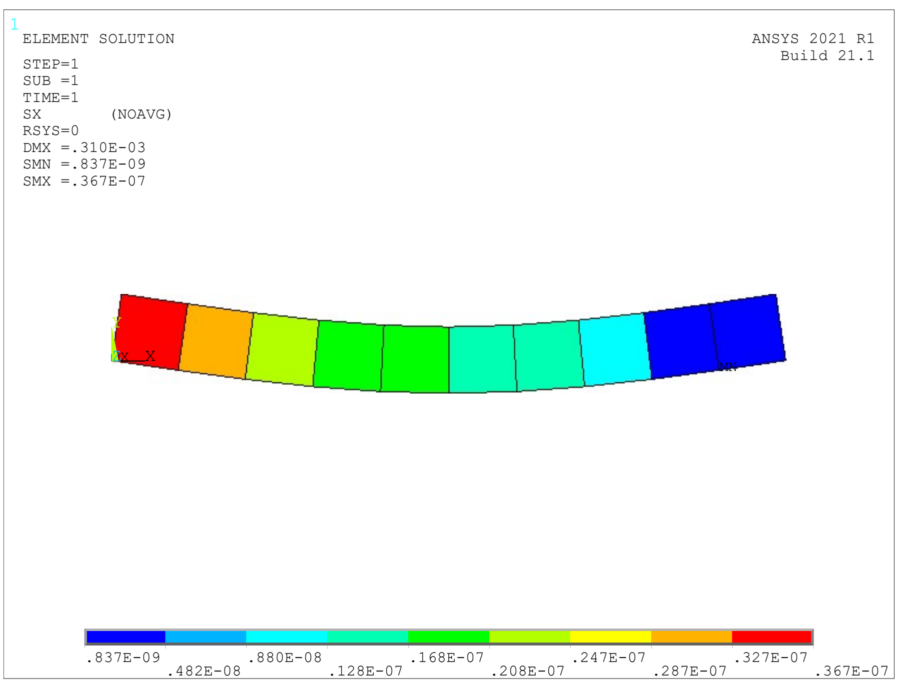

Figure 4: x stress contour plot of the 1x10 beam. (Units: Pa)

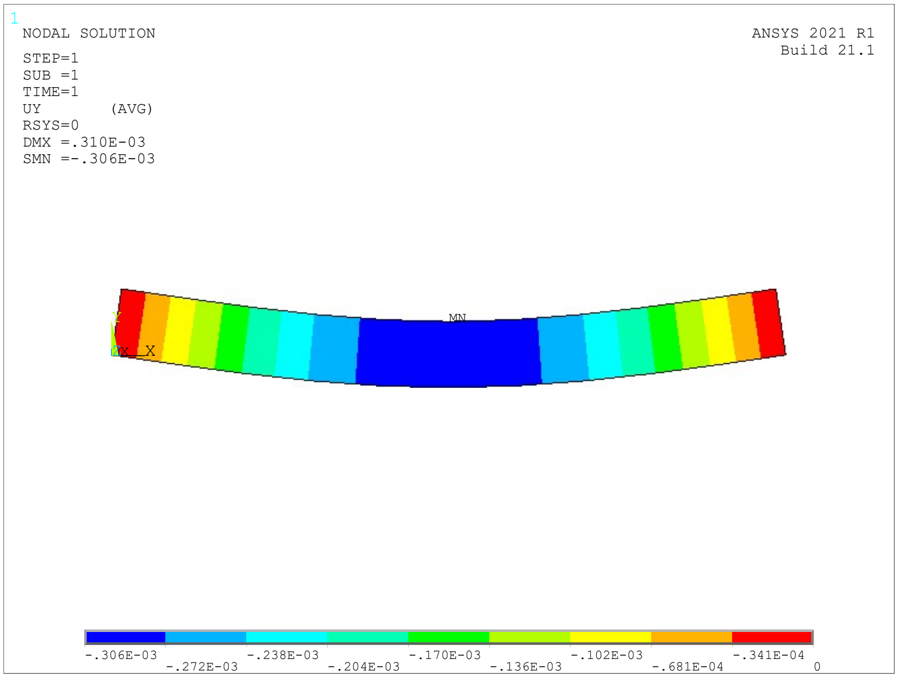

Figure 5: y displacement contour plot of the 1x10 beam. (Units: m)

The simulation calculated the same reaction forces as was found using the hand calculations. Outlined in Table 2 are the summarized simulation results.

What	Value
$\lvert\sigma_{max}\rvert$	$0.367 \times 10^{-7} Pa$
y_{max}	$0.306 \times 10^{-3} m$ downward

Table 2: Simulation stress and displacement results for the 1x10 beam.

Analysis

The stress in the bent beam modeled in this report is compressive at the top and tensile at the bottom. Because the mesh was only a single element deep, the stress at the top and bottom cancelled out resulting in the software computing no stress. The displacement also cannot be calculated accurately with this low of a mesh resolution, resulting in a calculation that is an order of magnitude off.

2x20

Results

The 2x20 resolution means the beam was meshed with 2 element in height and 20 elements along its length. Because the beam was modeled as 40 units long and 4 units tall, the 2x20 beam was meshed with elements that had edge lengths of 2 units (i.e. $2m$). This produced the mesh shown in Figure 6.

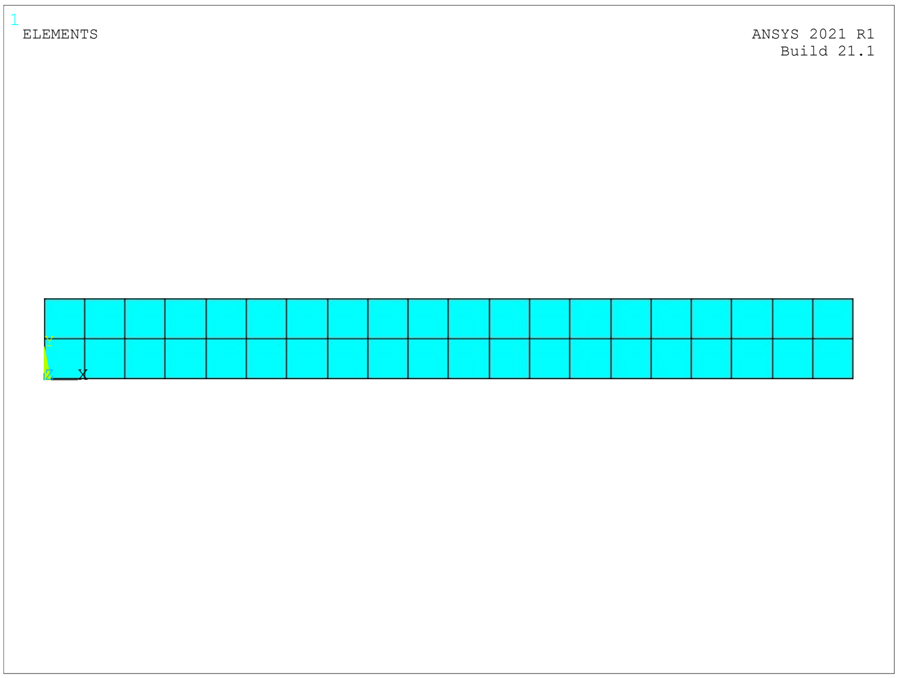

Figure 6: Mesh of the 2x20 beam.

Once simulated, the beam deforms as shown in Figure 7. The x stress and y displacement of this beam can be seen in Figures 8 and 9, respectively.

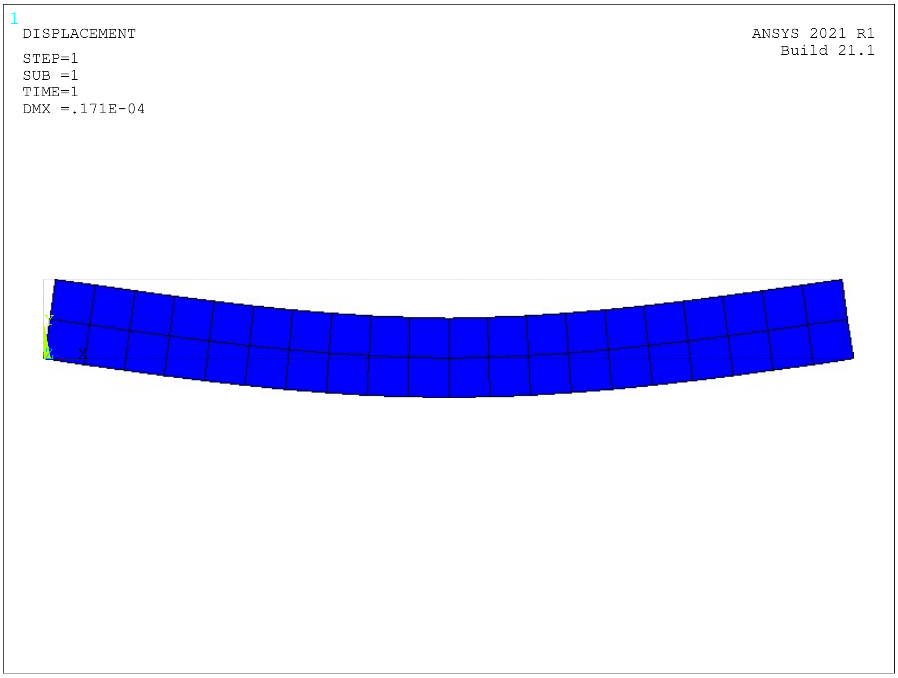

Figure 7: Deformed 2x20 beam.

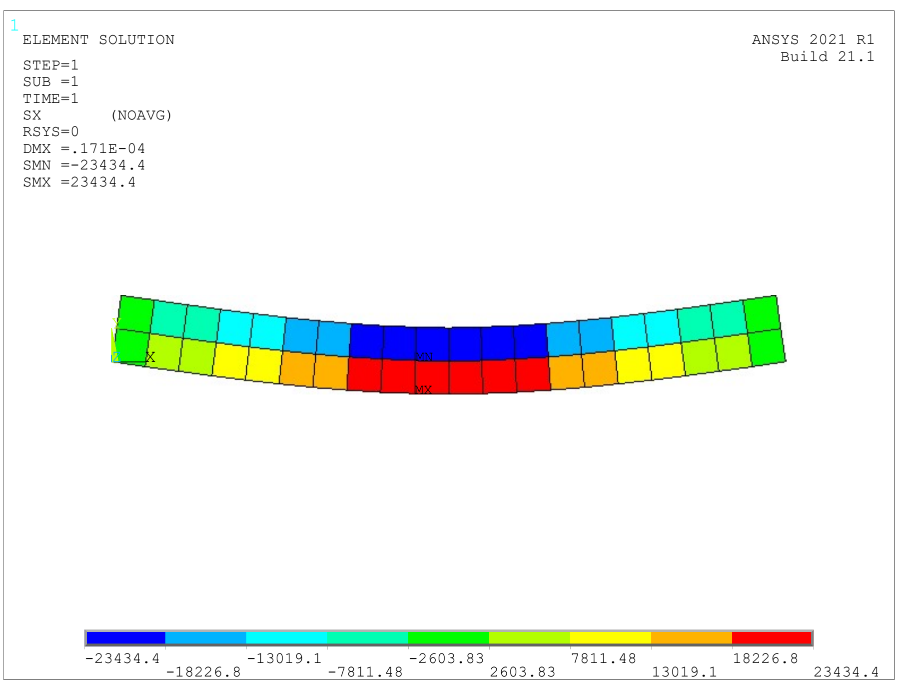

Figure 8: x stress contour plot of the 2x20 beam. (Units: Pa)

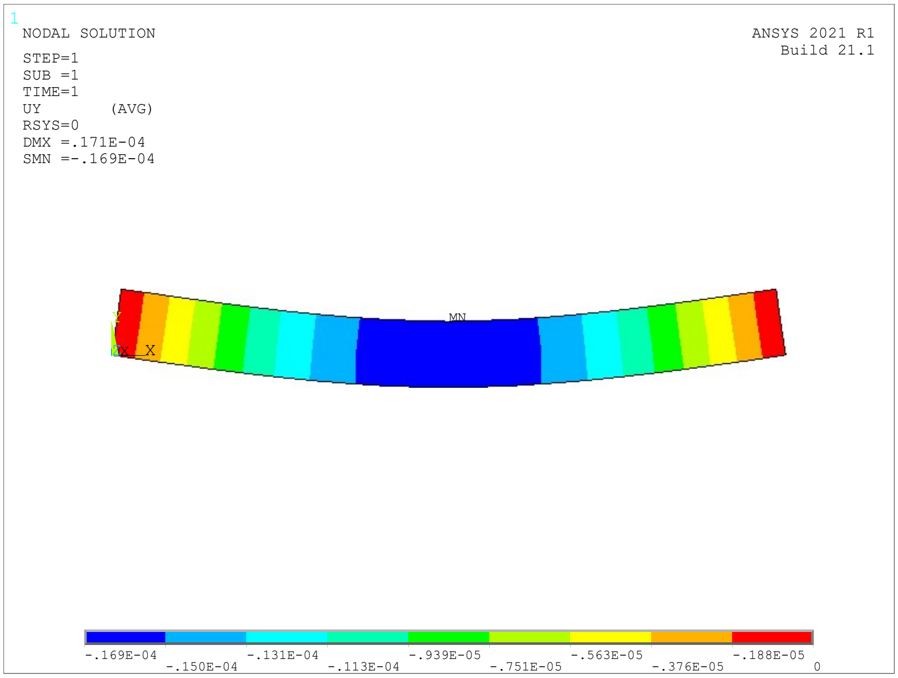

Figure 9: y displacement contour plot of the 2x20 beam. (Units: m)

The simulation calculated the same reaction forces as was found using the hand calculations. Outlined in Table 3 are the summarized simulation results.

What	Value
$\|\sigma_{max}\|$	$23434.4Pa$
y_{max}	$0.169 \times 10^{-4}m$ downward

Table 3: Simulation stress and displacement results for the 2x20 beam.

Analysis

With two elements in depth, the top and bottom compressive and tensile stresses can be resolved, so a stress value was calculated. Although the value is not exactly correct, it is still within an order of magnitude of the value calculated by hand. With this higher resolution, the displacement result is much more accurate and is very close to the hand-calculated value.

4x40

Results

The 4x40 resolution means the beam was meshed with 4 element in height and 40 elements along its length. Because the beam was modeled as 40 units long and 4 units tall, the 4x40 beam was meshed with elements that had edge lengths of 1 unit (i.e. $1m$). This produced the mesh shown in Figure 10.

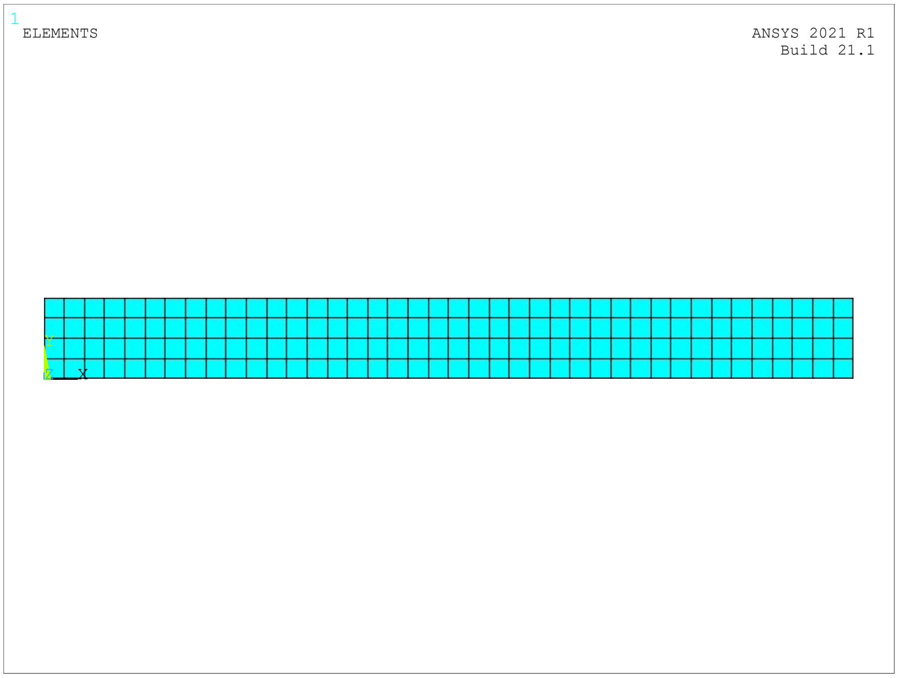

Figure 10: Mesh of the 4x40 beam.

Once simulated, the beam deforms as shown in Figure 11. The x stress and y displacement of this beam can be seen in Figures 12 and 13, respectively.

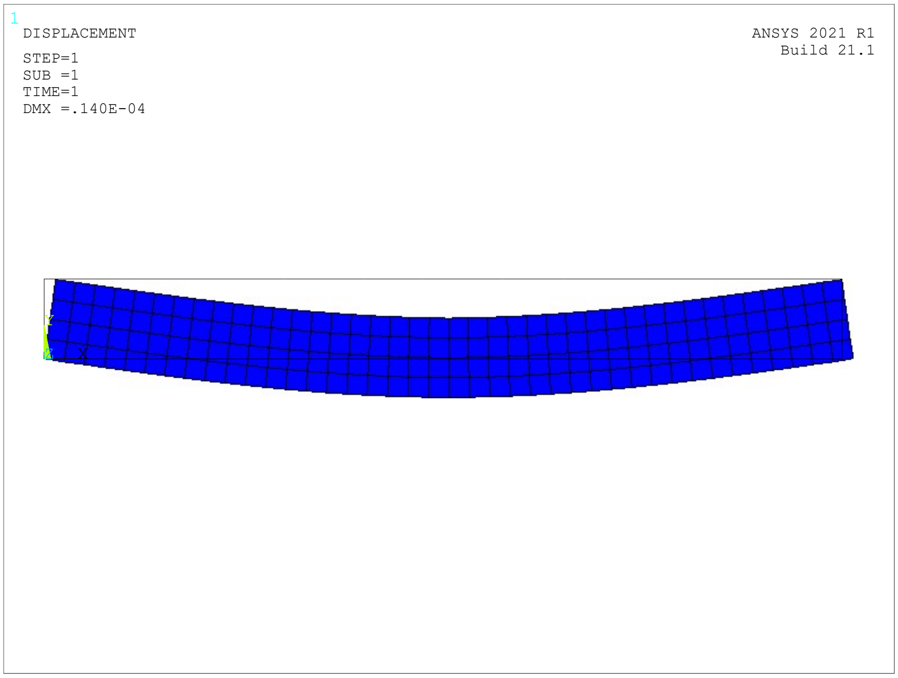

Figure 11: Deformed 4x40 beam.

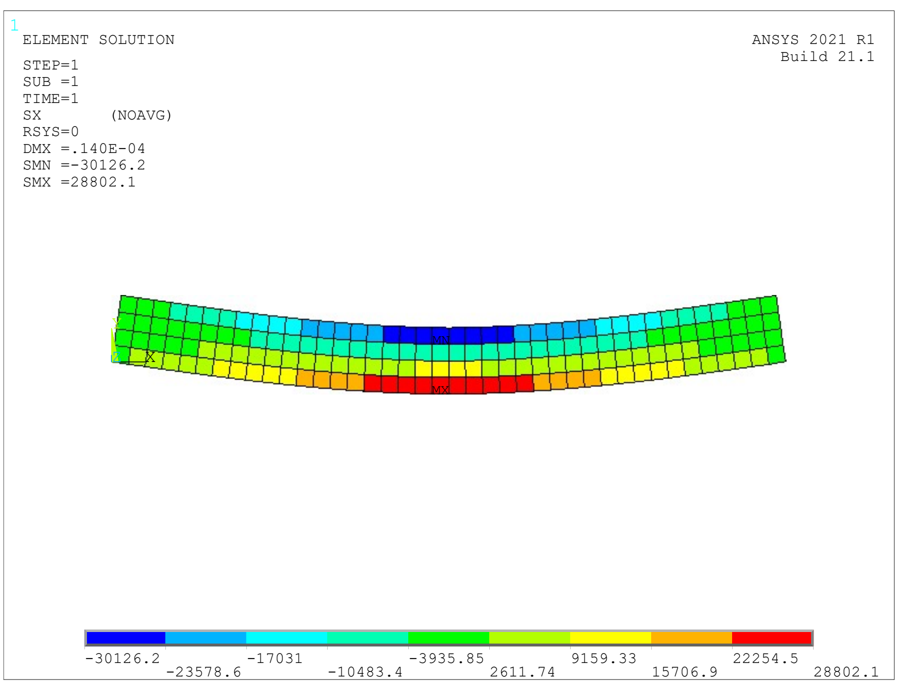

Figure 12: x stress contour plot of the 4x40 beam. (Units: Pa)

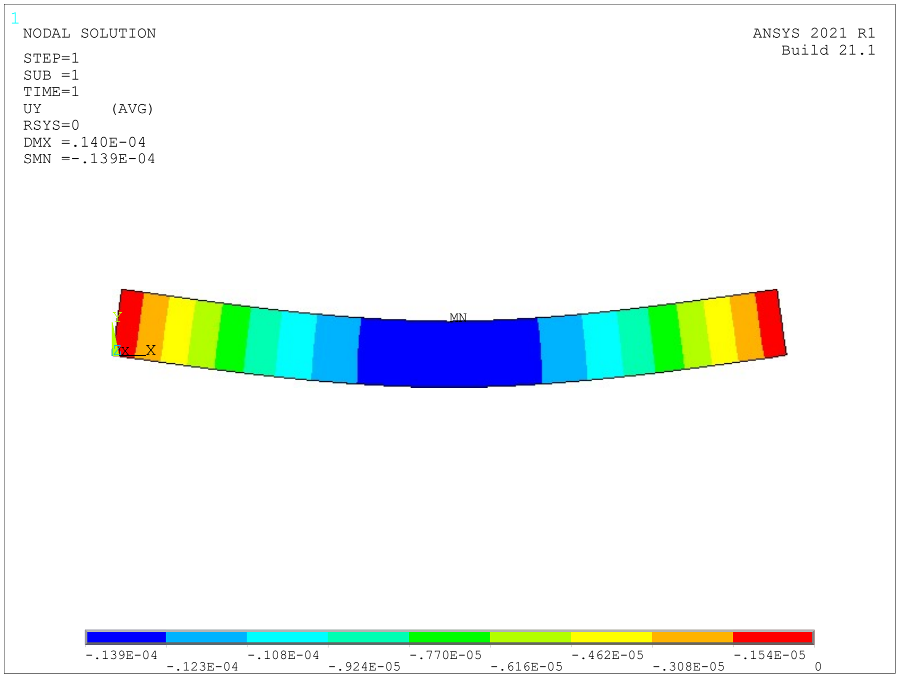

Figure 13: y displacement contour plot of the 4x40 beam. (Units: m)

The simulation calculated the same reaction forces as was found using the hand calculations. Outlined in Table 4 are the summarized simulation results.

What	Value
$\lvert\sigma_{max}\rvert$	$30126.2 Pa$
y_{max}	$0.139 \times 10^{-4} m$ downward

Table 4: Simulation stress and displacement results for the 4x40 beam.

Analysis

At the highest resolution investigated for this project, the stress and displacement values are very close to the values found through hand calculations. Increasing the resolution more will further improve the results until they converge on a single value.

Extra Credit: 1x4x40

Setup

As extra credit, a solid model was simulated. The simulation was performed using Ansys APDL finite element software (Version: 2021 R1 Build 21.1). The SOLID185 element with default settings was chosen to mesh the beam. SOLID185 is an 8-node brick element that has three degrees of freedom per node corresponding to three degrees of translation.[3] A structural, linear, elastic, isotropic material was chosen, and the Young's modulus and Poisson's ratio for steel was input.

The beam was modeled as a volume of 40 units long, 4 units tall, and 1 unit deep. The area was meshed with the solid elements of 1 unit edge length (i.e. $1m$ edge length); see the mesh in Figure 14. Because the loading is purely vertical (y), no out-of-plane motion should be present, so all nodes were restricted from displacing in the z direction. Additionally, the bottom left nodes were restricted from translating in the x or y directions and the bottom right nodes were restricted from translating in the y direction, modeling a pinned and roller support, respectively. To load the beam in an equivalent fashion as the beam being analyzed in the bulk of this report, the load was split between the two nodes at the middle of the top edge (i.e. $5kN$ downward load to the two nodes).

The simulation was run using the built-in default structural solver and settings in Ansys APDL.

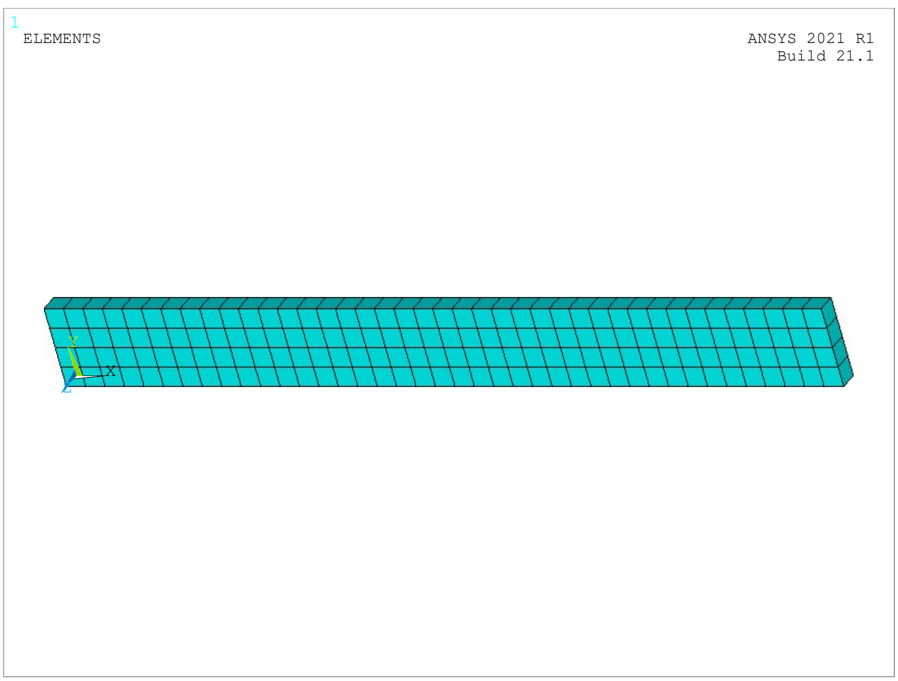

Figure 14: Mesh of the 1x4x40 beam.

Results

Once simulated, the beam deforms as shown in Figure 15. The x stress and y displacement of this beam can be seen in Figures 16 and 17, respectively.

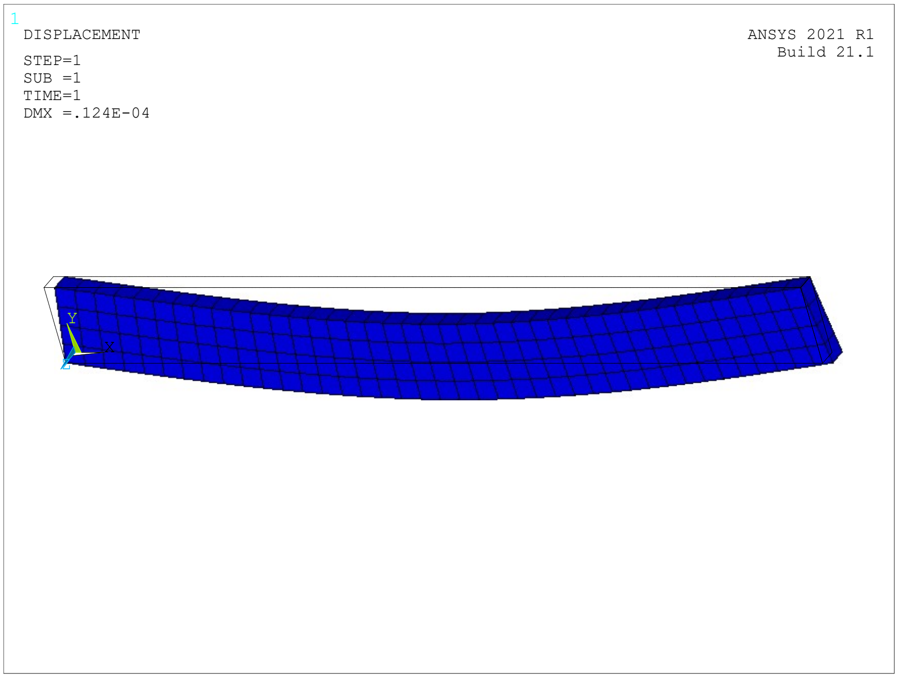

Figure 15: Deformed 1x4x40 beam.

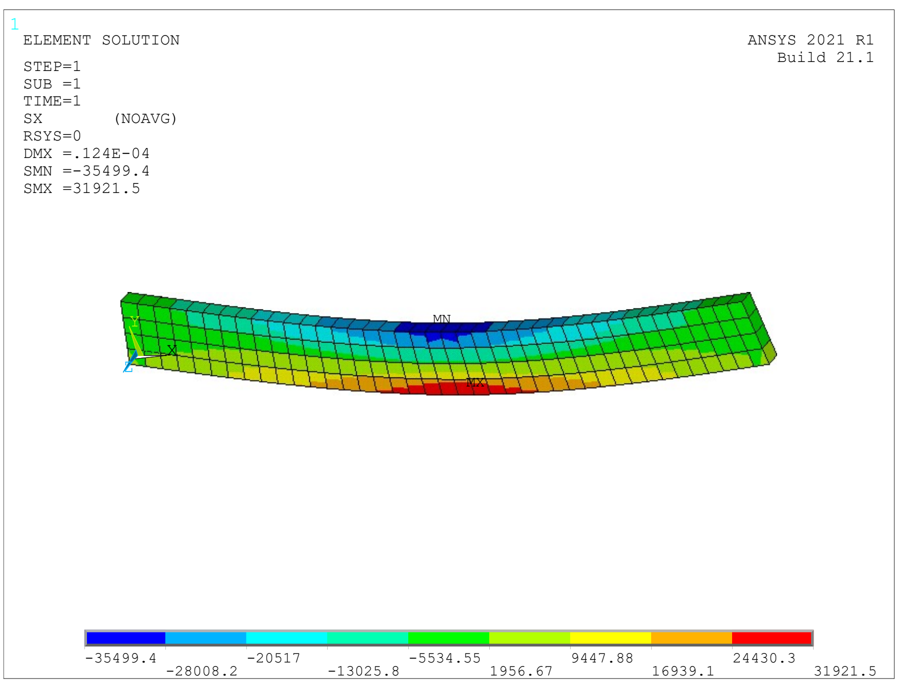

Figure 16: x stress contour plot of the 1x4x40 beam. (Units: Pa)

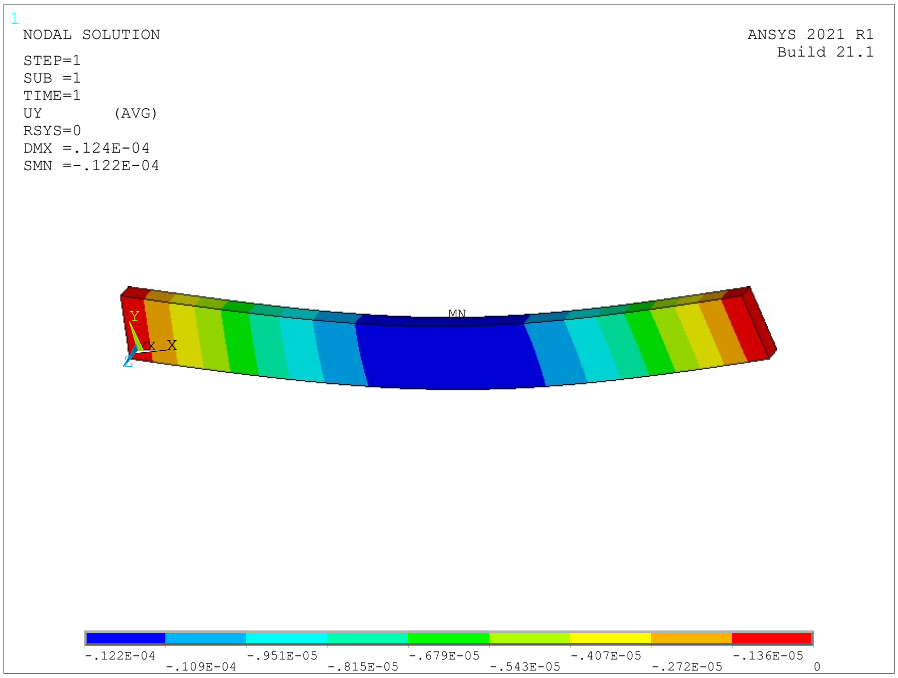

Figure 17: y displacement contour plot of the 1x4x40 beam. (Units: m)

The simulation calculated the same reaction forces as was found using the hand calculations, except the results were split between the now 4 nodes (i.e. each of the four supported nodes had a reaction force of $2.5kN$). Outlined in Table 5 are the summarized simulation results.

What	Value
$\|\sigma_{max}\|$	$35499.4Pa$
y_{max}	$0.124 \times 10^{-4}m$ downward

Table 5: Simulation stress and displacement results for the 1x4x40 beam.

Analysis

Compared to the 4x40 model meshed with shell elements, which was the 2D model that had the closest results to the hand calculations, the solid model produced results even closer to the hand calculations. This can probably be attributed to the solid elements more accurately modeling the inherently 3D beam. Although it is more accurate, it is slightly more complicated to setup, making a high-resolution 2D model that still produces sufficiently accurate results more appealing.

Conclusion

Overall, the simply supported beam was simulated in Ansys APDL. The simulations were verified through hand calculations. The low resolution beam, i.e. 1x10, could not resolve the stress due to the extremely low mesh resolution and was an order of magnitude off from the values calculated by hand. The higher resolution beams, i.e. 2x20 and 4x40, produced good results. Running the simulations at different resolutions shows the importance of having a high enough resolution that the physics can accurately be modeled. If successive simulations were to further increase the resolution, the results would eventually converge on a solution. Because the simulations were extremely quick to run at the resolutions investigated in this report, it would be a simple task to increase the resolution significantly, e.g. 40x400. A 3D 1x4x40 model produced more accurate results for the same number of elements as the 4x40 model, but this was at the cost of more complexity when setting up the simulation, making the increased-resolution for the 2D model still an appealing option.

Appendix

Detailed Hand Calculations

The displacement, moment, and reaction forces equations taken from *Marks' Handbook* are shown below.[4]

$$R_A = R_B = -\frac{P}{2} \tag{1}$$

$$M(x) = \frac{P}{2}x \tag{2}$$

$$y_{max} = \frac{PL^3}{48EI} \tag{3}$$

where I is the moment of inertia of the cross section. For a rectangle, the equation is as follows.

$$I = \frac{1}{12}bh^3 \tag{4}$$

where b is the width ($1m$ for this case) and h is the height ($4m$ for this case).

The stress in the beam can be calculating using the stress for pure bending.[5] Although this is not a pure bending situation, it is a good approximation. The equation for the maximum stress in the beam is shown below.

$$\sigma_{max} = \frac{Mc_{max}}{I} \tag{5}$$

where c_{max} is the maximum distance from the neutral axis, which is the beam axis that does not experience any strain.

Plugging in the $10kN$ downward load into Equation (1), the reaction forces were both found to be $\boxed{5kN}$ upwards. The highest moment would be found at the center of the beam, so, plugging in $x = \frac{L}{2}$, the moment was found to be $100kNm$. The moment of inertia was calculated to be $\frac{16}{3}m^4$.

At this point, all the necessary values are known to calculate the maximum displacement. Plugging the values in yields a maximum displacement of $\boxed{0.125 \times 10^{-4}m}$ downward.

To calculate the maximum stress, the neutral axis needs to be known. For a rectangular cross-sectioned beam, the neutral axis is at half the height, so $c_{max} = 2m$ since it is at most $2m$ from the neutral axis to the furthest point on the beam (in height). Plugging in the calculated values yields a maximum stress of $\boxed{37500Pa}$.

APDL Inputs and Outputs

Simulation settings used for all 2D mesh resolutions.

Software	Ansys APDL version: 2021 R1 Build 21.1
Model	Structural
Material	Linear>Elastic>Isotropic
Young's Modulus	200GPa
Poisson's Ratio	0.25
Element Type	SHELL181 (default settings)
Layup Thickness	1 (1 layer)
Mesh Size	1x10: 4 2x20: 2 4x40: 1
Bottom Left	0 displacement in x, y, and z 0 rotation in x and y
Bottom Right	0 displacement in y and z 0 rotation in x and y
Remaining Nodes	0 displacement in z 0 rotation in x and y
Loads	10kN downward at the middle of the top surface
Solver	Default structural solver

Table 6: Specific APDL settings used to develop the simulations for all 2D the mesh resolutions.

2D nodal reaction forces.

Node	Force [N]
Bottom Left y	5000.0
Bottom Left x	0.0
Bottom Right y	5000.0

Table 7: All mesh resolutions' reaction forces.

Simulation settings used for the 3D model.

Software	Ansys APDL version: 2021 R1 Build 21.1
Model	Structural
Material	Linear>Elastic>Isotropic
Young's Modulus Poisson's Ratio	200GPa 0.25
Element Type	SOLID185 (default settings)
Mesh Size	1 unit
Bottom Left Bottom Right Remaining Nodes	0 displacement in x, y, and z 0 displacement in y and z 0 displacement in z
Loads	5kN downward at the 2 middle nodes of the top surface
Solver	Default structural solver

Table 8: Specific APDL settings used to develop the 3D simulation.

Python Code

```python
# -*- coding: utf-8 -*-
"""
Jared Jacobowitz
Fall 2021
CE422 Finite Element Method
Project 2

Code to calculate the reaction forces, moment, stress, and displacement
"""

def moment(P, x):
    """Moment in a simply supported beam"""
    return P*x/2

def moment_of_inertia(b, h):
    """Moment of inertia of a rectangle"""
    return b*h**3/12

def displacement(P, L, x, E, I):
    """Displacement of simply supported beam loaded at the center"""
    return -P*x*(3*L**2 - 4*x**2)/(48*E*I)

def stress(M, c, I):
    """Stress in a beam under pure bending"""
    return M*c/I

def main():
    E = 200e9           # Pa; Young's modulus steel
    L = 40              # m; length of the beam
    h = 4               # m; height of the beam
    b = 1               # m; depth of the beam
    P = -10e3           # N; applied force

    x = L/2             # m; location of applied force

    M_max = moment(P, x)
    I = moment_of_inertia(b, h)
    delta_max = displacement(P, L, x, E, I)
    stress_max = stress(M_max, h/2, I)

    print(f"{M_max=} Nm, {I=} m^4, {delta_max=} m, {stress_max=} Pa")

if __name__ == "__main__":
    main()
```

References

[1] *Steel Material Properties*. URL: http://www.matweb.com/search/datasheet.aspx?bassnum=MS0001&ckck=1.

[2] Ansys Inc. *SHELL181*. July 2017. URL: https://www.mm.bme.hu/~gyebro/files/ans_help_v182/ans_elem/Hlp_E_SHELL181.html.

[3] Ansys Inc. *SOLID185*. July 2017. URL: https://www.mm.bme.hu/~gyebro/files/ans_help_v182/ans_elem/Hlp_E_SOLID185.html.

[4] Ali M Sadegh. "5-2 Mechanics of Materials". In: *Marks' Standard Handbook for Mechanical Engineers*. 11th ed. McGraw-Hill Professional Publishing, 2006, pp. 5–22.

[5] *Stresses & Deflections in Beams*. URL: https://mechanicalc.com/reference/beam-analysis.