J2 VON-MISES PLASTICITY WITH APPLIED ELEMENT METHOD

Serdar MERMER[1], Serkan BEKİROĞLU[2]

Bursa Technical University, Bursa, Turkey
Yıldız Technical University, İstanbul, Turkey

***Abstract**: Applied Element Method (AEM) is a relatively new and advanced numerical analysis method by which structural behavior from elastic range to total collapse can be simulated. AEM combines the advantages and capabilities of Finite Element Method (FEM) and Discrete Element Method (DEM). The structural system is modeled as an assemblage of small rigid body elements with this method. The neighboring elements are connected by pairs of zero-length normal and shear springs. In this study, J2 Von-Mises Plasticity with AEM is presented. Some 2D numerical models including steel beams are generated, and the analysis results are compared with FEM results. It can be said that AEM could be used for numerical analysis of elasto-plasticity for structural systems with appropriate mesh and connections successfully.*

***Key words**: Applied Element Method (AEM), Finite Element Method (FEM), Rigid Body, Shear and Normal Spring, J2 Von-Mises Plasticity, Steel Beams*

1. Introduction

The Applied Element Method (AEM) [1] was introduced around the late 90s as a new method to be able to simulate total behavior of structures from beginning to collapse**.** Nonlinear analysis widely has been done mostly all types of structural systems using the Finite Element Method (FEM), but after the AEM is introduced, this can also be done alternatively with the latter.

This paper introduces the implementation of one of the most widely used plasticity, especially on metallic materials, von-Mises plasticity with AEM. The next sections in this paper briefly, gives some generic information of AEM, applying von-Mises plasticity to the AEM analysis. Some numerical examples including 2-D cantilever steel beam, unstiffened and stiffened steel beams are analyzed, and the comparison results are presented.

[1] Serdar MERMER, PhD Candidate, Department of Civil Engineering, Bursa Technical University, Bursa, Turkey, serdar.mermer@btu.edu.tr
[2] Serkan BEKİROĞLU, Prof. Dr, Ph.D., Faculty of Civil Engineering, Department of Civil Engineering, İstanbul, Turkey, serkanb@yildiz.edu.tr

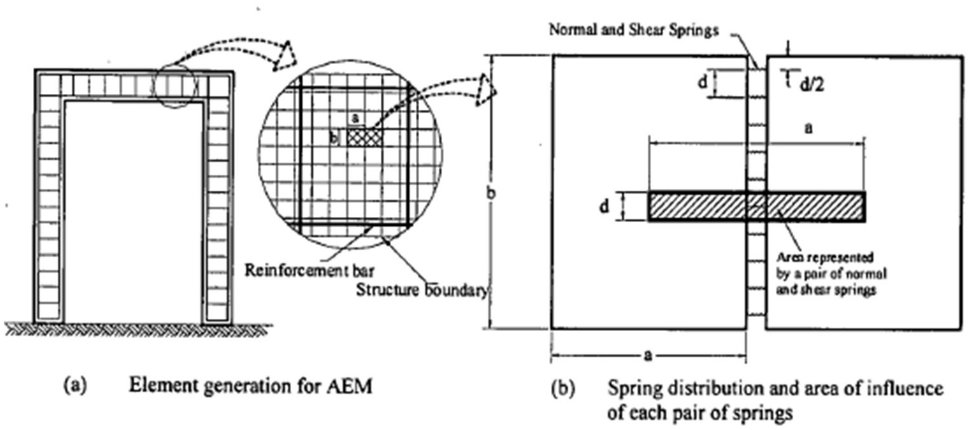

Figure 1. Modeling of Structures in AEM [1]

2. Overview of AEM

In AEM all structural domains consist of rigid body elements that are connected with pair of normal and shear springs across the adjacent edges of the elements. The springs are being used to define the stresses and strains in that area.

2.1. Formulation of AEM

The degree of freedom is described in the center of the elements. For a 2-D element three degrees of freedom are defined for one element (Fig. 2). The stiffness matrix for a 2-D element, which is 6 x 6 matrix, for one pair of elements by applying unit displacement to the related degree of freedom and by calculating reaction forces while rest of DOFs are supported.

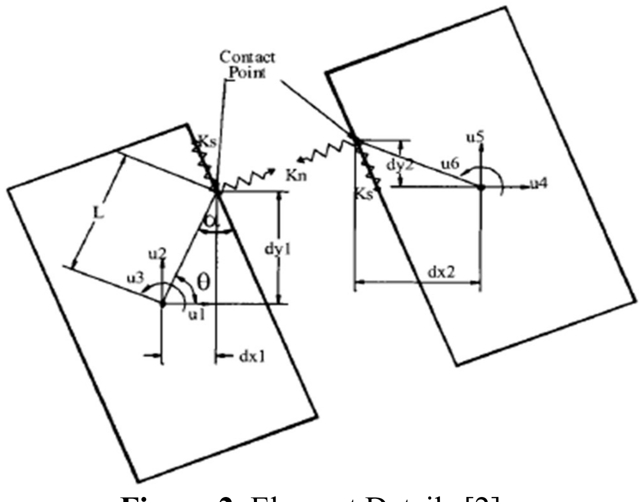

Figure 2. Element Details [2]

The upper left quarter of the stiffness matrix is displayed in Fig. 3.

$$
\begin{bmatrix}
\begin{array}{c|c|c}
Sin^2(\theta+\alpha)K_n + Cos^2(\theta+\alpha)K_s & -K_n Sin(\theta+\alpha)Cos(\theta+\alpha) + K_s Sin(\theta+\alpha)Cos(\theta+\alpha) & Cos(\theta+\alpha)K_s L Sin(\alpha) - Sin(\theta+\alpha)K_n L Cos(\alpha) \\
\hline
-K_n Sin(\theta+\alpha)Cos(\theta+\alpha) + K_s Sin(\theta+\alpha)Cos(\theta+\alpha) & Sin^2(\theta+\alpha)K_s + Cos^2(\theta+\alpha)K_n & Cos(\theta+\alpha)K_n L Cos(\alpha) + Sin(\theta+\alpha)K_s L Sin(\alpha) \\
\hline
Cos(\theta+\alpha)K_s L Sin(\alpha) - Sin(\theta+\alpha)K_n L Cos(\alpha) & Cos(\theta+\alpha)K_n L Cos(\alpha) + Sin(\theta+\alpha)K_s L Sin(\alpha) & L^2 Cos^2(\alpha)K_n + L^2 Sin^2(\alpha)K_s
\end{array}
\end{bmatrix}
$$

Figure 3. Upper left quarter of the stiffness matrix [2]

The spring stiffnesses K_n and K_s are defined as below equation Eq. (1),

$$\boldsymbol{K_n} = \frac{Edt}{a}\ ;\ \boldsymbol{K_s} = \frac{Gdt}{a} \tag{1}$$

where, d is the distance between two springs, t is the thickness of the element, a is the length of the hatched area (Fig. 1b), E and G are the elasticity and shear modulus of the material, respectively. For each element all the pairs of spring stiffness matrices are assembled to obtain the global stiffness matrix.

3. J2 Von-Mises Plasticity with AEM

In the general definition of von Mises criterion [3], plastic yielding starts when the J2 stress deviator invariant reaches a critical value. Mathematically the von Mises yield criterion is written as below Eq. (2),

$$\boldsymbol{\sigma_y} = \sqrt{3J_2} \tag{2}$$

where, σ_y is the tensile of yield strength of the material.

3.1. Applying J2 Von-Mises Plasticity to AEM

In the traditional plastic analysis in Finite Element Method (FEM) approach, all calculations related to the plasticity mostly are done on the gauss point level. On the other hand, in AEM all calculations related the plasticity must be done on the for each pair of springs.

4. Numerical Examples

In this section, two numerical examples are investigated. All AEM results are obtained from our software that is being developed using Microsoft Visual C# [4] and Eyeshot which is a 3rd party CAD software component [5]. FEM results are obtained from Ansys WB software [6].

4.1. Plasticity of an End-Loaded Cantilever Beam

In this example, plasticity of an End-Loaded cantilever beam (Fig. 4) with a rectangular section is being analyzed [7]. Comparison results between AEM and FEM are presented. The limit load is found to be around 30 kN as in the reference book. A perfectly plastic von-Mises material model using Plane Stress Analysis is considered. During the analysis, Force control is adopted.

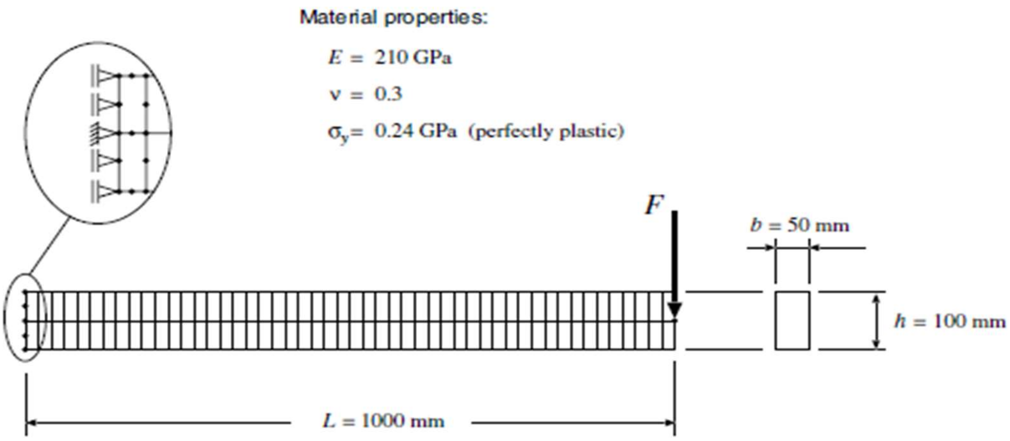

Figure 4. Geometry of End-Loaded Cantilever Beam [7]

Results of the converged solution for both AEM and FEM in the last increment are presented below. Displacement contours for vertical direction in AEM and FEM are introduced in Fig. 5, Fig. 6. All displacement contours indicate that both AEM and FEM show very good agreement.

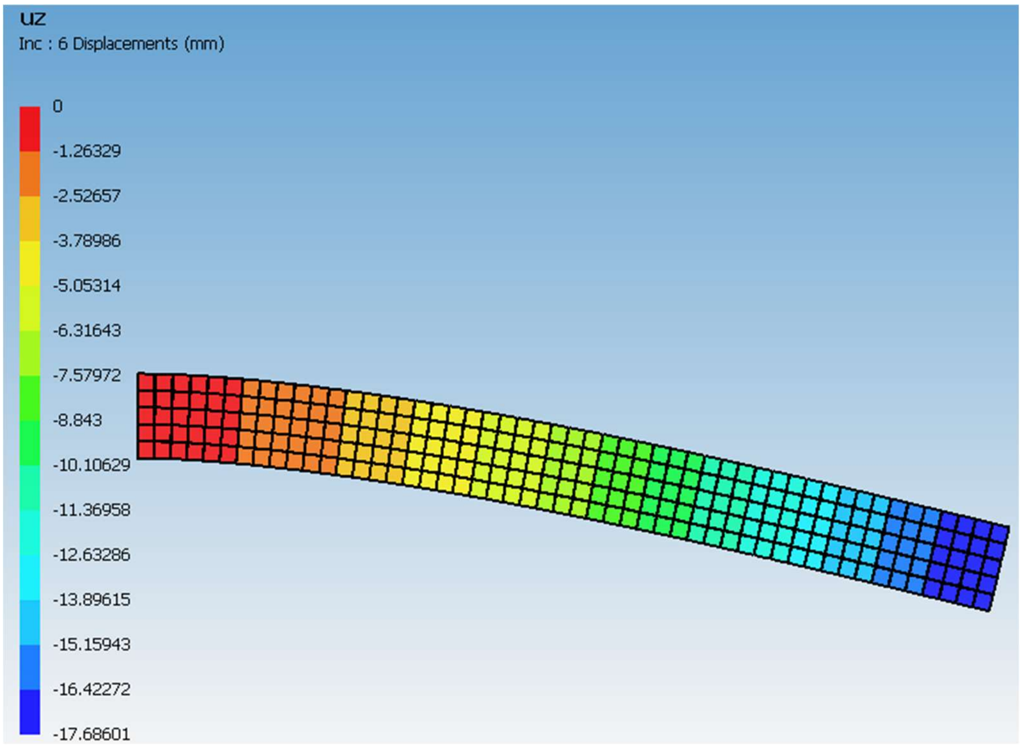

Figure 5. Vertical displacement contour in AEM

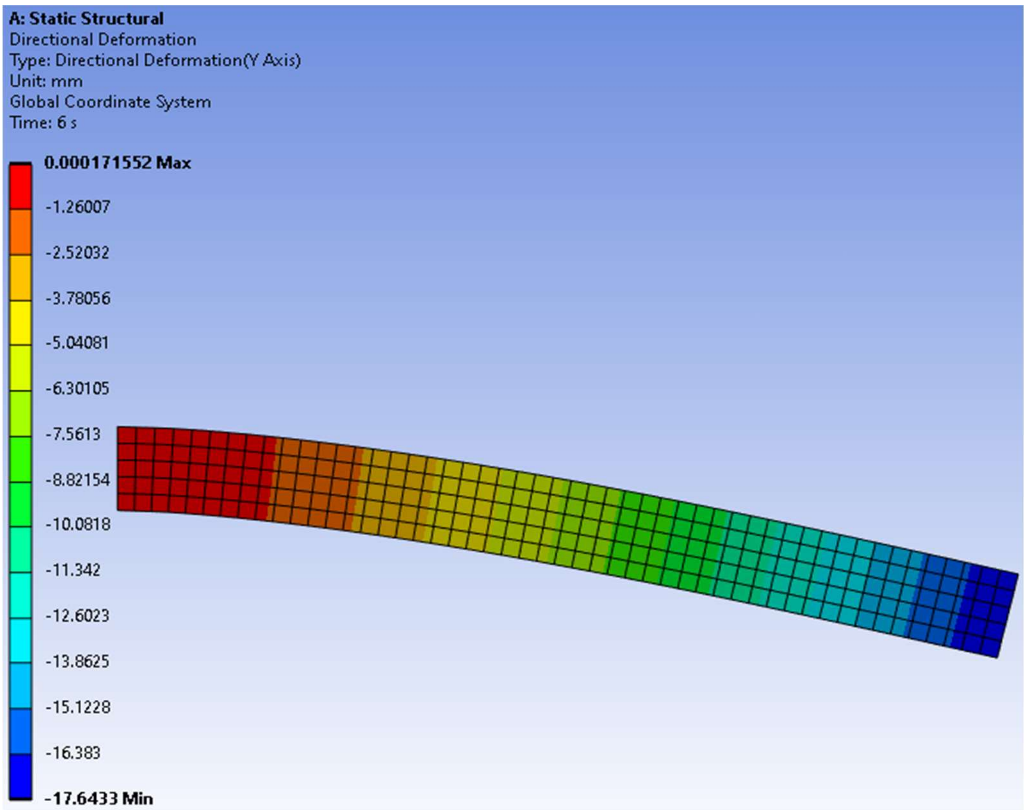

Figure 6. Vertical displacement contour in FEM (Ansys)

To be able to see the plastic behavior during the analysis, below Applied Load vs Vertical displacement chart at loaded node is presented (Fig. 7). From the chart, it can be said that after the incrementation of 30 kN, the behavior of plasticity begins.

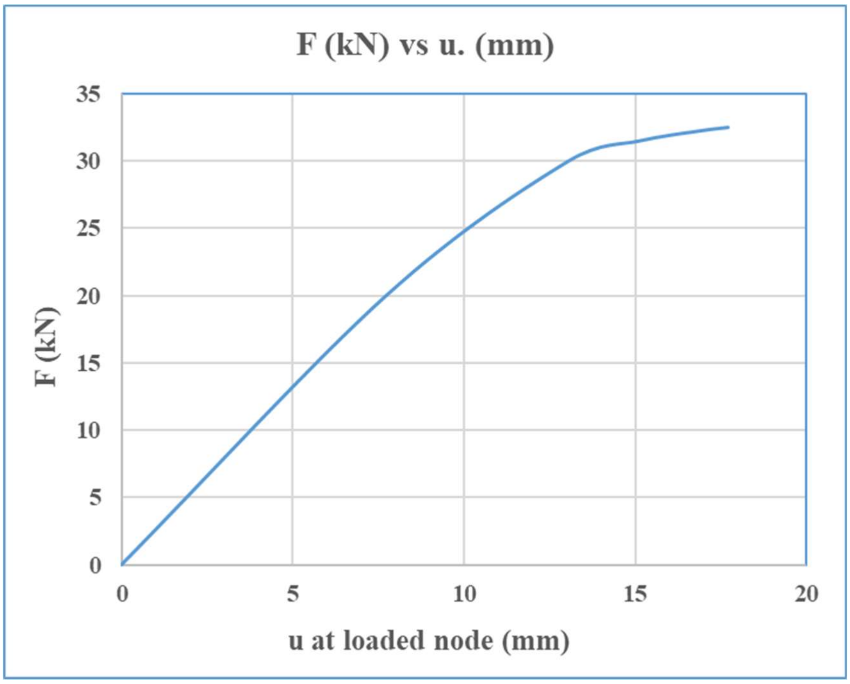

Figure 7. Load-Displacement curve

The other parameter that is important for plastic analysis is accumulated equivalent plastic strain contours that show which part of the elements is plastic, are presented for AEM and FEM (Fig. 8, Fig. 9).

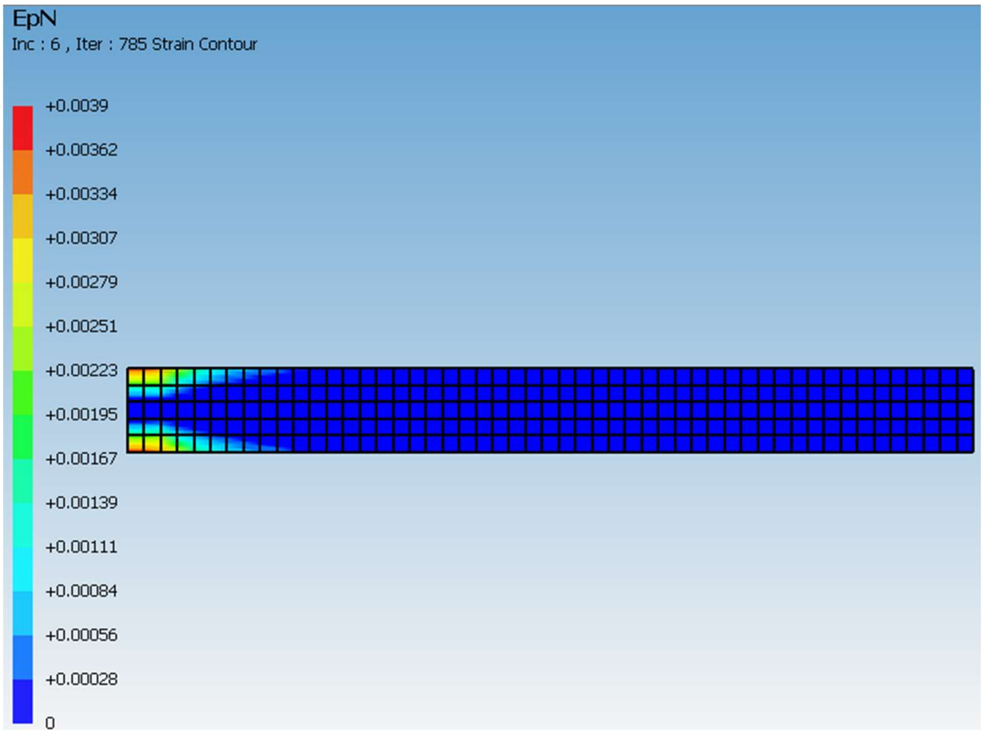

Figure 8. Accumulated equivalent plastic strain contour in AEM

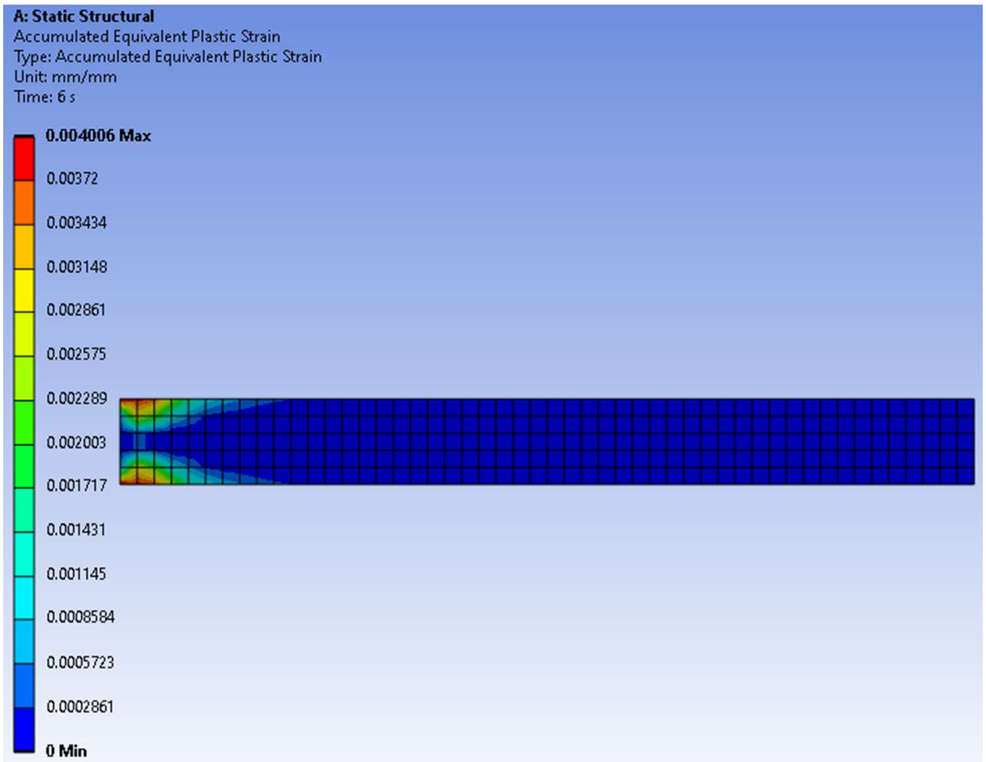

Figure 9. Accumulated equivalent plastic strain contour in FEM (Ansys)

The same parameter distribution near the supported section of the beam is shown below in Fig. 10. It can be said that accumulated equivalent plastic strains are increasing linearly from near the center of the section to the far locations of the beam section as expected.

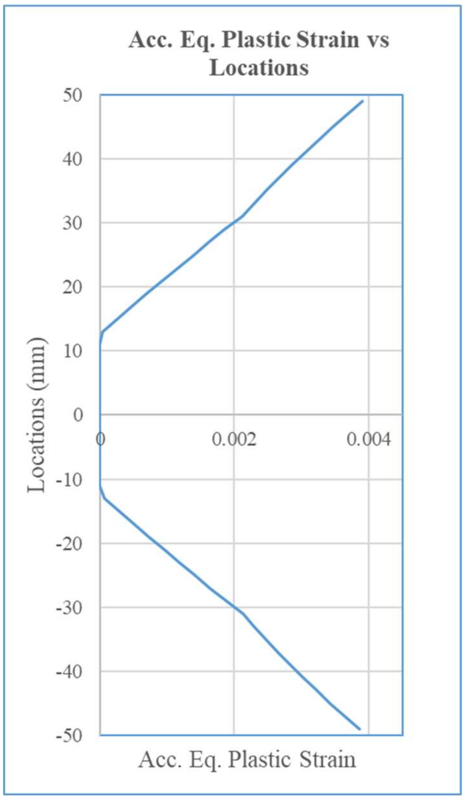

Figure 10. Accumulated equivalent plastic strains distribution across the supported section

4.2. Plasticity of a w/wo Stiffened Steel Girder

In this example, plasticity of with and without stiffened steel girders with an I section are being analyzed (Fig. 11). Some comparison results between AEM and FEM are presented. A bi-linear plastic von-Mises material model using Plane Stress Analysis is considered. During the analysis, force control is adopted and applied load F is around 150 kN.

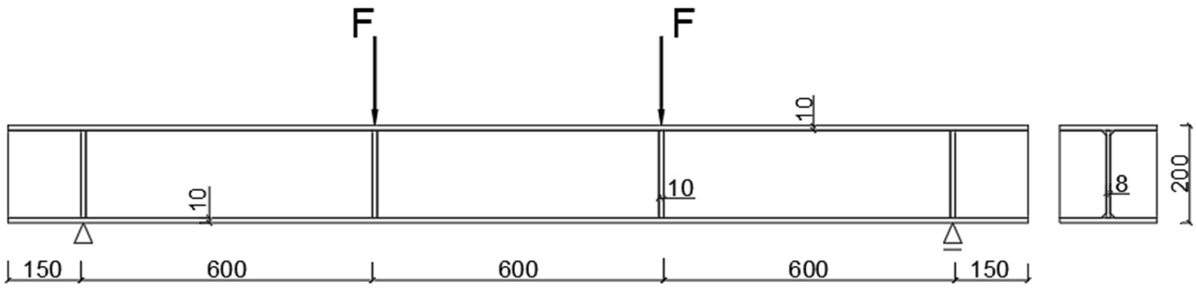

Figure 11. Stiffened Steel Girder Detail (all units are mm)

Some results of the converged solutions for both AEM and FEM in the last increment are presented below. Firstly, unstiffened steel girder results are presented. Displacement contours for vertical direction in AEM and FEM are introduced in Fig. 12, Fig. 13. All displacement contours indicate that both AEM and FEM show good agreement.

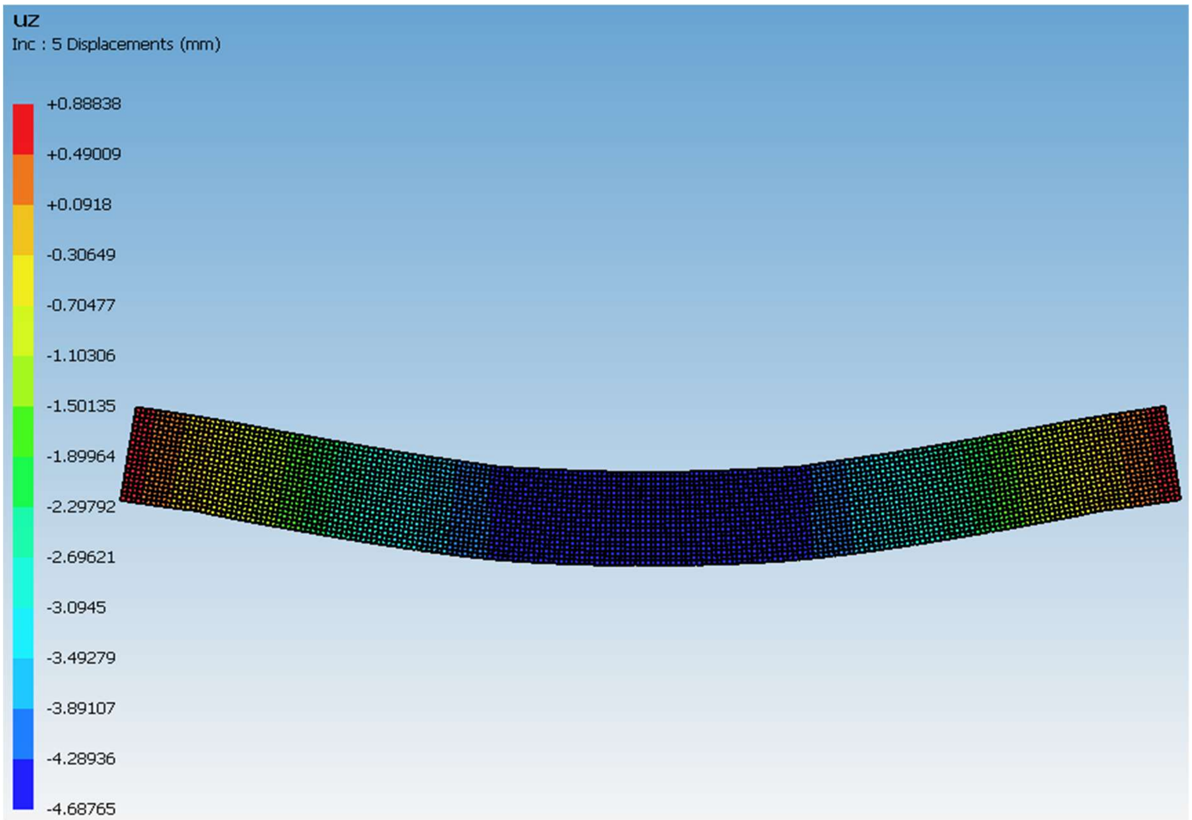

Figure 12. Vertical displacement contour for unstiffened steel girder in AEM

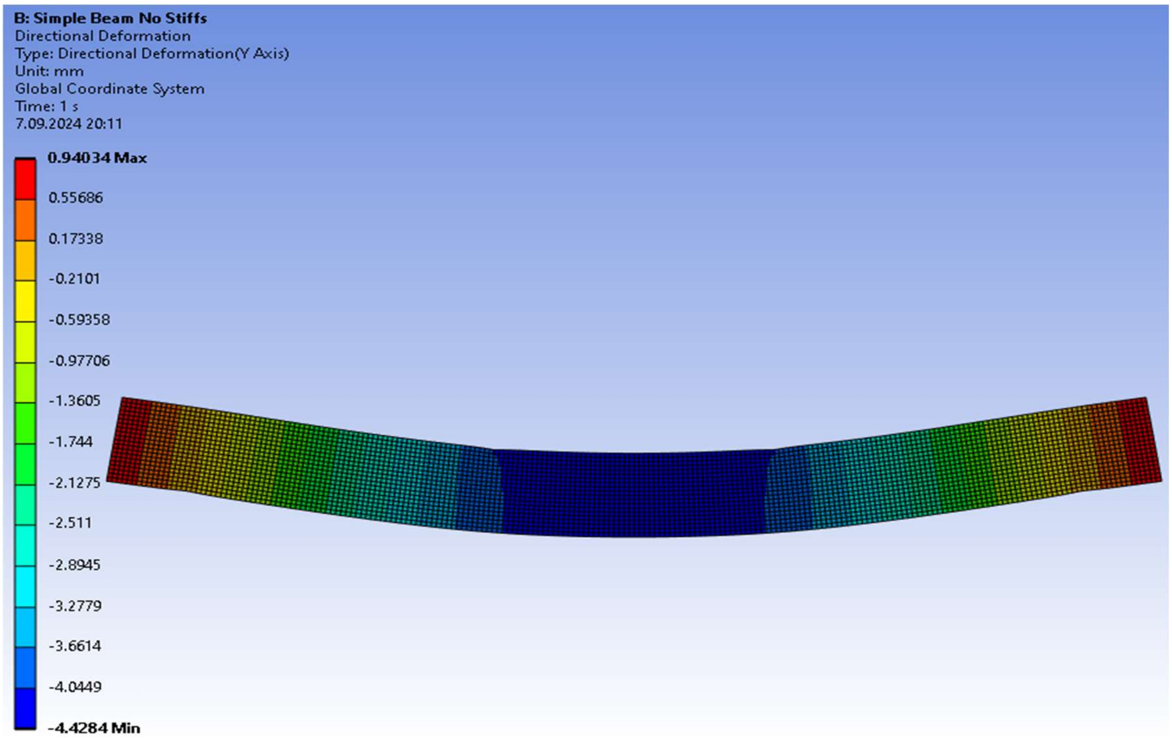

Figure 13. Vertical displacement contour for unstiffened steel girder in FEM (Ansys)

The other parameter that controls which part of the elements are plastic when exceeding the yielding stress during the plastic analysis is Von-Mises stress, also called Equivalent stress contours, are presented for AEM and FEM (Fig. 14, Fig. 15). As can be seen from both figures, near both support and loading zones, some elements are being plastic which means at the same time, their equivalent stress values are found beyond yielding stress.

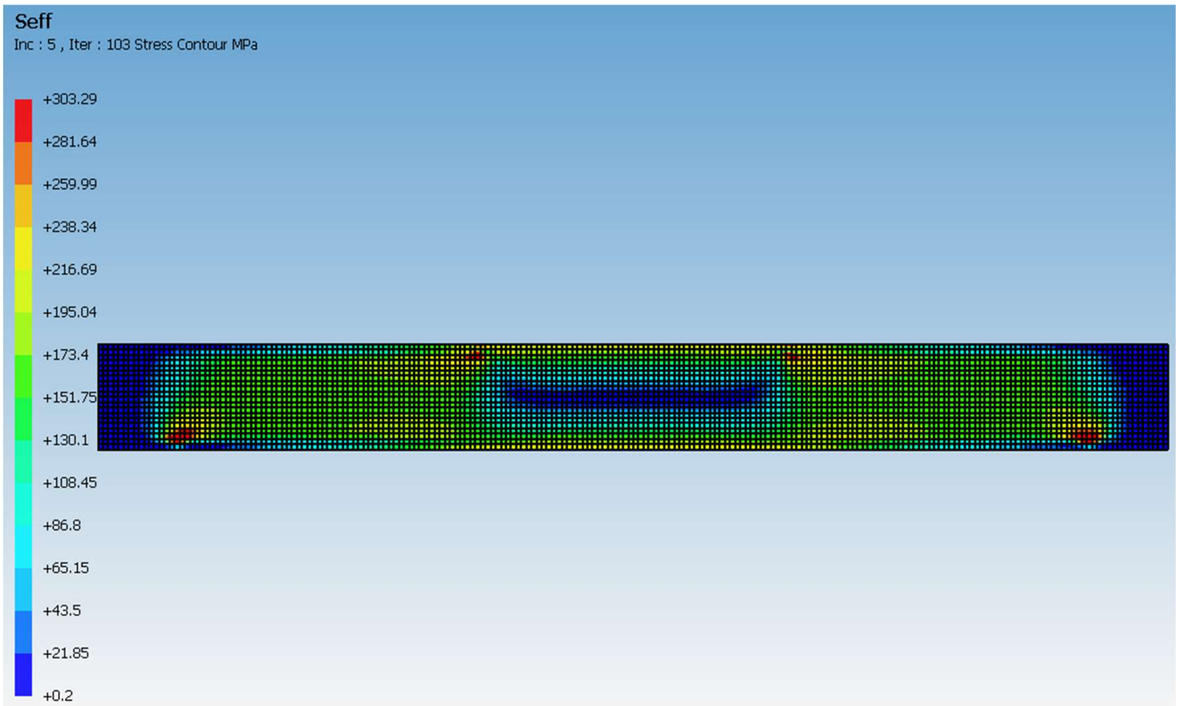

Figure 14. Von-Mises stress contour for unstiffened steel girder in AEM

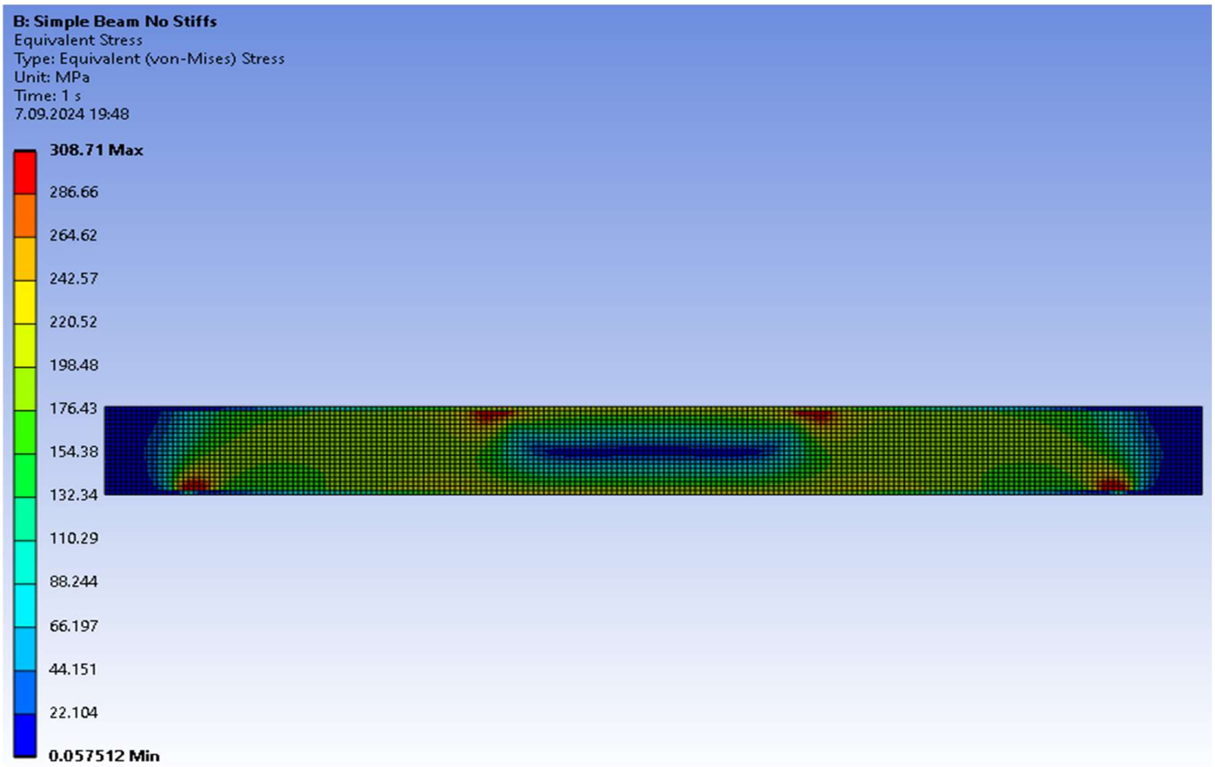

Figure 15. Von-Mises stress contour for unstiffened steel girder in FEM (Ansys)

From both figures (Fig. 16, Fig. 17) it can be said that stiffeners have changed stress distribution and decreased the stress values around the critical support and loading zones.

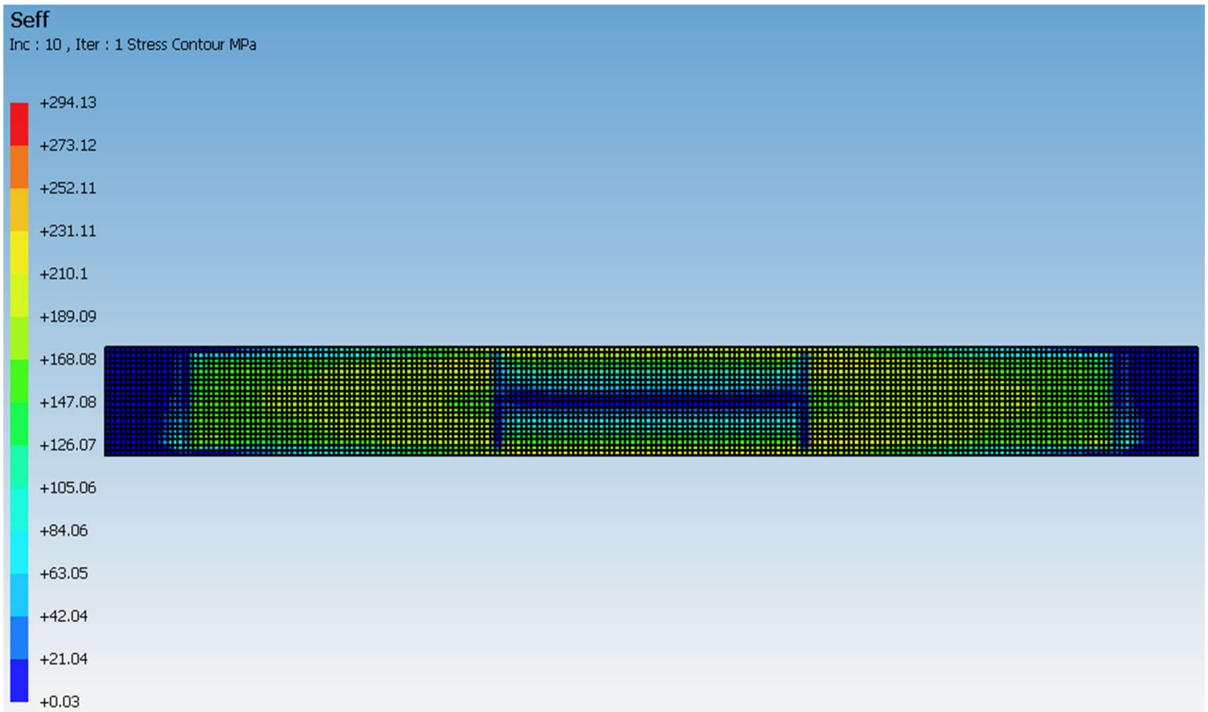

Figure 16. Von-Mises stress contour for stiffened steel girder in AEM

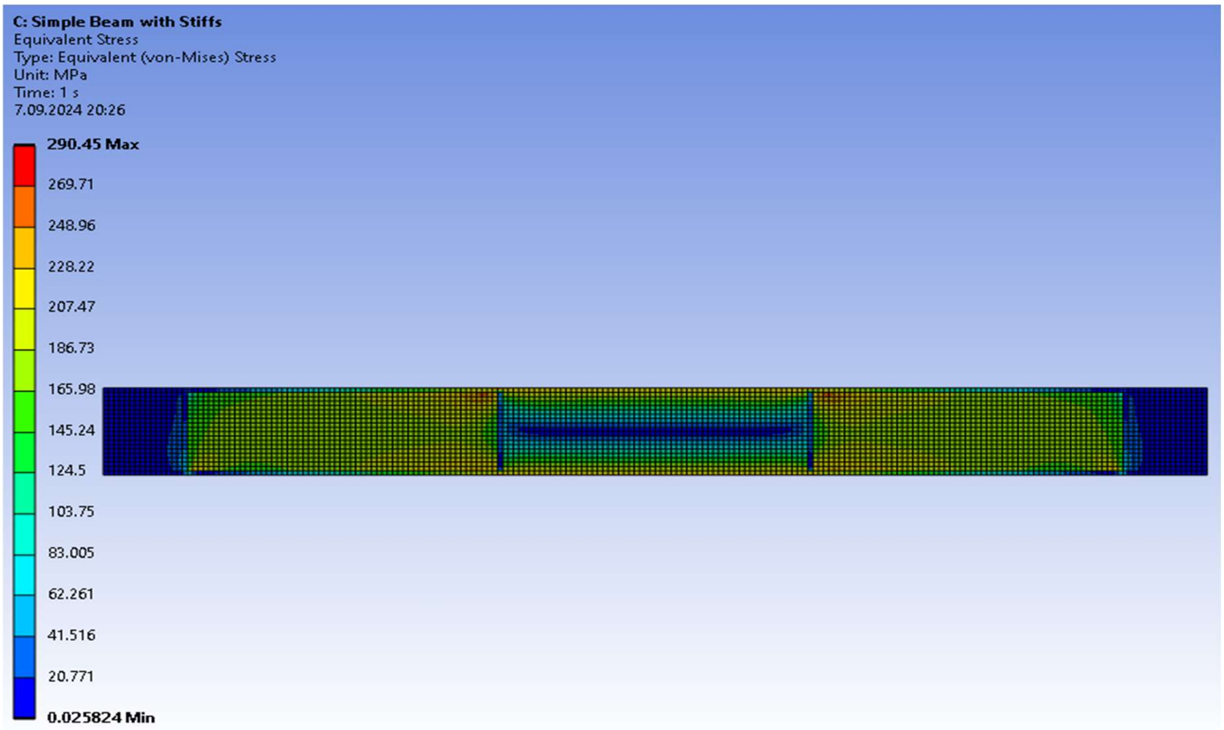

Figure 17. Von-Mises stress contour for stiffened steel girder in FEM (Ansys)

Conclusion

The conclusion of the study presented as follows; the Applied Element Method is also easily applicable when modeling plasticity in structures and giving very promising results as the alternatives as well. Future works, including the implementation of 3D plasticity and other types of material plasticity, are planned.

Acknowledgement

This work was supported by the project no “2016/05.01.DOP07 of YTU Coordinatorship of Scientific Research Projects”.

REFERENCES

[1] Tagel-Din, H., and Meguro, K. (2000). ‘‘Nonlinear simulation of RC structures using applied element method.’’ Struct. Engrg./EarthquakeEngrg., 17(2), 137s-148.

[2] Meguro, K., and Tagel-Din, H. (2000). ‘‘Applied element method for structural analysis: Theory and application for linear materials.’’ Struct.Engrg./Earthquake Engrg., 17(1), 21s–35s.

[3] von Mises, R. (1913). "Mechanik der festen Körper im plastisch-deformablen Zustand". *Nachrichten von der Gesellschaft der Wissenschaften zu Göttingen*. Mathematisch-Physikalische Klasse. **1913** (1): 582–592

[4] Microsoft, *Microsoft C#*, Version 11.0, Microsoft Corporation, 2023. [Online]. Available: https://www.microsoft.com/dotnet/csharp

[5] Eyeshot, Version 2024.2.342, devDept Software S.r.l., [Online]. Available: https://www.devdept.com/

[6] ANSYS Workbench, Version 2024, ANSYS Inc., 2024. Available: https://www.ansys.com

[7] Eduardo A de Souza Neto, Djordje Peric, and David RJ Owen. Computational methods for plasticity: theory and applications. John Wiley & Sons, 2011.