

Analysis of Welded Connection Plates by Using ANSYS (Comparative Study – Finite Element Analysis with Elastic Analysis)

B. Vamsi Krishna¹; T. Shanthala²; P. Sudheer Kumar³; S. Bhoomesh⁴

¹Assistant Professor, Department of Civil Engineering, Malla Reddy Engineering College (Autonomous), Secunderabad, India.

²Assistant Professor, Department of Civil Engineering, Srinivasa Ramanujan Institute of Technology, Ananthapur, India.

³Assistant Professor, Department of Civil Engineering, Balaji Institute of Technology and Science, Narsampet, Warangal, India.

⁴Assistant Professor, Department of Civil Engineering, Malla Reddy Engineering College (Autonomous), Secunderabad, India.

Abstract

Structural analysis of the loaded welded connection of plates requires the determination of the design strength of the plate specially in bridge Structures. Simple welded shear tab connections are used regularly in structural engineering practice. Many engineers who still use Allowable Stress Design use the elastic method in designing these connections. To determine the stresses in a plate at the welded joint of a simple shear tab connection using finite element analysis, and to compare those results to the simplified elastic analyses used in structural engineering practice. In particular we will look at the tension and compression stresses at the extreme fibers of the weld connection along with the shear stress at the middle of the weld. ANSYS will be used for the finite element analysis. In conclusion, both the finite element analysis approach and the allowable stress design approach result in almost identical maximum shear stresses in the plate at the welded joint. The two approaches do not provide similar results for the maximum tension and compression stresses at the ends of the joint.

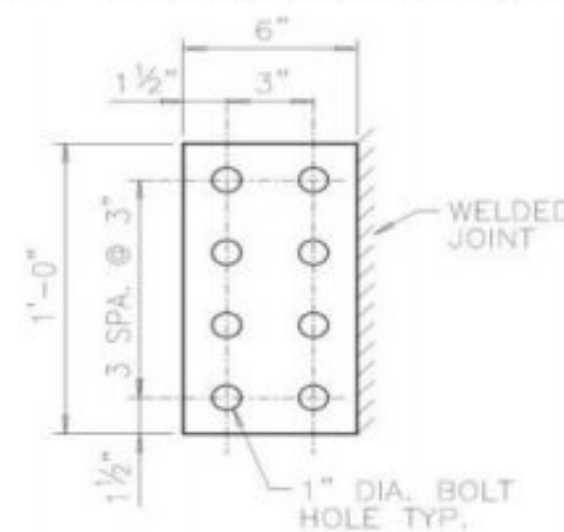
Key-words: ANSYS, Finite Element Analysis, Elastic Analyses, Welded Connections.

1. Introduction

Simple welded shear tab connections are used regularly in structural engineering practice. Many engineers who still use Allowable Stress Design use the elastic method in designing these connections. This method is spelled out in a number of design references [1]; a very commonly used

one being the American Institute of Steel Construction (AISC) manual. Since this type analysis limits the capacity of the material to its elastic range, a linear elastic material type will be used in ANSYS. The material properties that will be used will be of A36 steel as defined in the American Society of International Association for Testing and Materials (ASTM) standards [2]. This steel has an elastic modulus of 29,000 ksi and a Poisson's Ratio of 0.26. The geometry of the connection plate under Investigation is shown in Figure 1.

Figure 1 - Connection Plate Geometry



All of the ANSYS models created for this project will be in 2D and plane stress will be used. This is appropriate for thin plate type analyses.

2. Objective

Our project objective is to determine the stresses in a plate at the welded joint of a simple shear tab connection using finite element analysis, and to compare those results to the simplified elastic analyses used in structural engineering practice. In particular we will look at the tension and compression stresses at the extreme fibers of the weld along with the shear stress at the middle of the weld. ANSYS will be used for the finite element analysis.

3. Verification

To verify the ANSYS model is behaving correctly, a check will be done prior to adding the bolt holes. The goal of this check is to determine if the material properties and boundary conditions are producing results that can be reasonably verified through calculations. These material properties and boundary conditions are what will be used in the final plate arrangement [3]. The check will be to verify the vertical displacement at the free end of the simple plate computed by ANSYS. The material properties are described in the previous section (Background). As for the boundary conditions, the right edge of the plate will be set to zero displacement in all degrees of freedom. Once convergence

of this simple plate model is achieved (Appendix A – Simple Plate Model Convergence), vertical point loads will be applied to all of the nodes. This is done in an effort to simulate an equivalent downward area load applied to the face of the plate. The equivalent area load will be what is used for the verifications calculations. The mesh that finally converged had a total of 433 nodes. A vertical load of -0.2 kips was applied to each of the nodes resulting in a maximum deflection of 3.1×10^{-3} inches at the free end of the plate. Figure 2 shows the results of this ANSYS analysis. The equivalent area load to be used for the verification calculations is shown below.

$$A = \text{Area Load} = \frac{N \cdot P}{L \cdot H} = \frac{(433 \text{ Nodes}) (0.2 \text{ kip})}{(6 \text{ in})(12 \text{ in})} = 1.203 \text{ ksi}$$

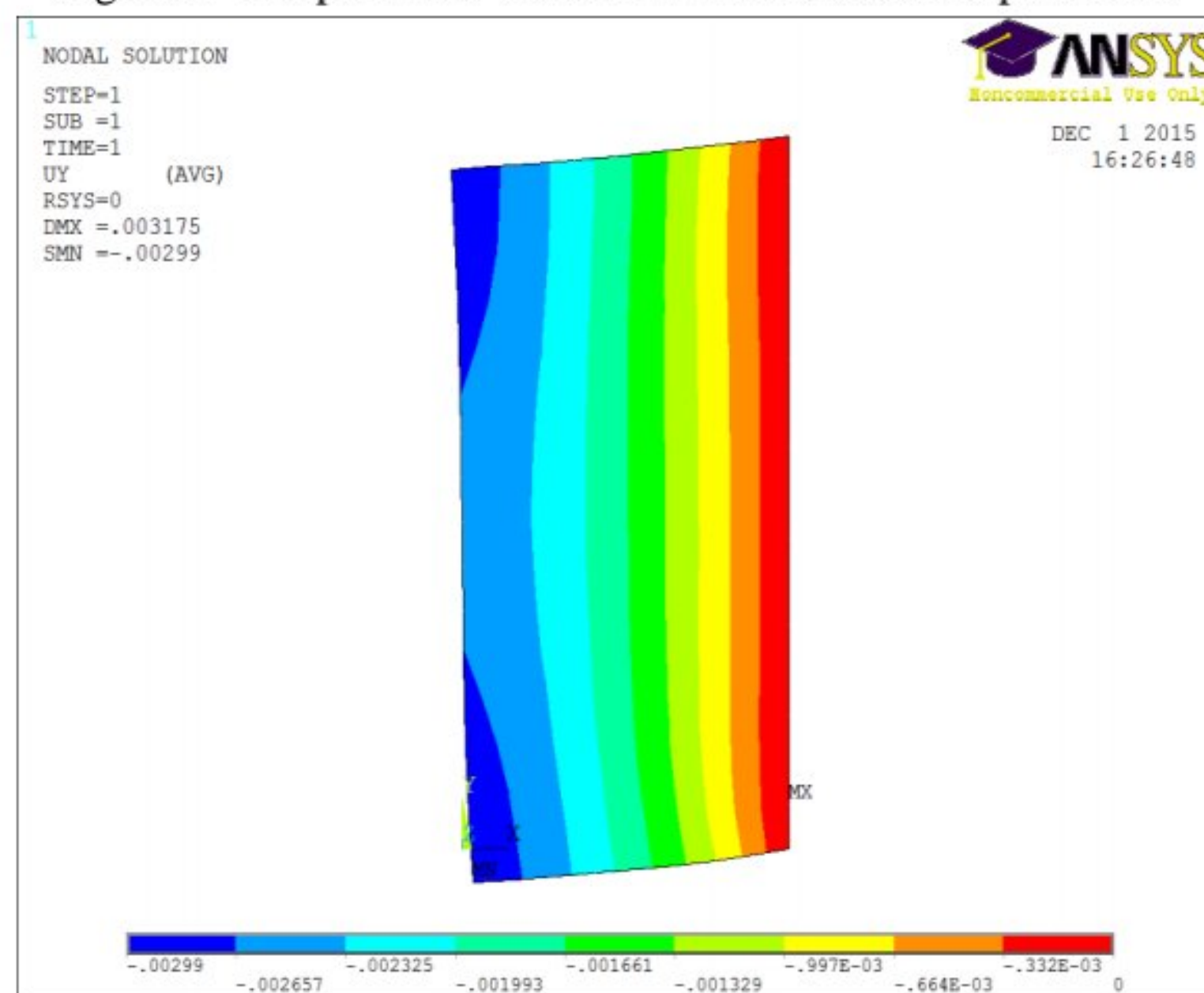
$N = \text{Number of Nodes} = 433 \text{ Nodes}$

$P = \text{Force Per Node} = 0.2 \text{ kip}$

$L = \text{Length of Connection Plate} = 6 \text{ in}$

$H = \text{Height of Connection Plate} = 12 \text{ in}$

Figure 2 - Simple Plate - ANSYS Vertical Nodal Displacement



To verify the deflection computed by ANSYS, we need to derive an equation for the deflection of connection plate due to a vertical area load applied to the face of the plate. We can relate the shear stress in the plate at any point along the length to the applied area load by the following equation:

$$\tau = A \cdot x$$

Where x equals 0 at the left side of the plate and L at the right side. We can then relate the shear stress to the engineering shear strain with shear modulus (G).

$$\gamma = \frac{\tau}{G} = \frac{A \cdot x}{G}$$

If we integrate the shear strain over the length of the connection plate, we can arrive at the theoretical displacement of the plate at the free end. The verification calculations result in a smaller vertical displacement at the free end of the plate. But that is expected because the verification calculations don't take into account displacement due to the slight bending of the plate. Therefore we conclude the model is behaving as expected and we can move on to the more complicated analysis of the bolted connection

4. Final ANSYS Model

After determining the material properties and boundary conditions were working correctly, the bolt holes were added and the connection plate was meshed until reaching convergence of the maximum shear stress (Appendix B – Final Model Convergence). The element type used was the plane stress 6-node triangle. The reason for using the 6-node triangle instead of the 8-node quad was to help eliminate issues with overly distorted element shapes. The loads applied were linear loads of 10 k/in to the bottom half of the 8 bolt holes. The loads were applied perpendicular to the perimeter of the 1 inch diameter bolt holes. However, the total vertical force per bolt is equivalent to the 10 k/in linear load multiplied by the diameter of the bolt, so 10 kips per bolt. This will be important in comparing the ANSYS results to elastic analyses used in practice. See proof below.

$$\Sigma F_y = \int_0^{\pi r} l \cdot \sin(r) dx = l \int_0^{\pi r} \sin(r) dx = l \cdot 2r = l \cdot d$$

l = linear load (10 k for this case)

r = radius of bolt hole

d = diameter of bolt hole

There will be some local effects around the bolt holes due to the horizontal components of the loads acting perpendicular to the perimeter of the holes. However, they will be equal and opposite for each bolt. For the purpose of this project, we will assume these loads will resolve locally at each bolt and have negligible effects on the stresses at the locations along the welded joint that are being investigated.

5. ANSYS Results

The ANSYS analysis produced a maximum shear stress of approximately 9.992 ksi at the middle of the joint (See Figure 3). It also produced a maximum tension stress of 31.7 ksi at the top of the joint, and a maximum compression stress of 35.9 ksi at the bottom of the joint (See Figure 4).

Figure 3 - Simple Plate - ANSYS Vertical Nodal Displacement

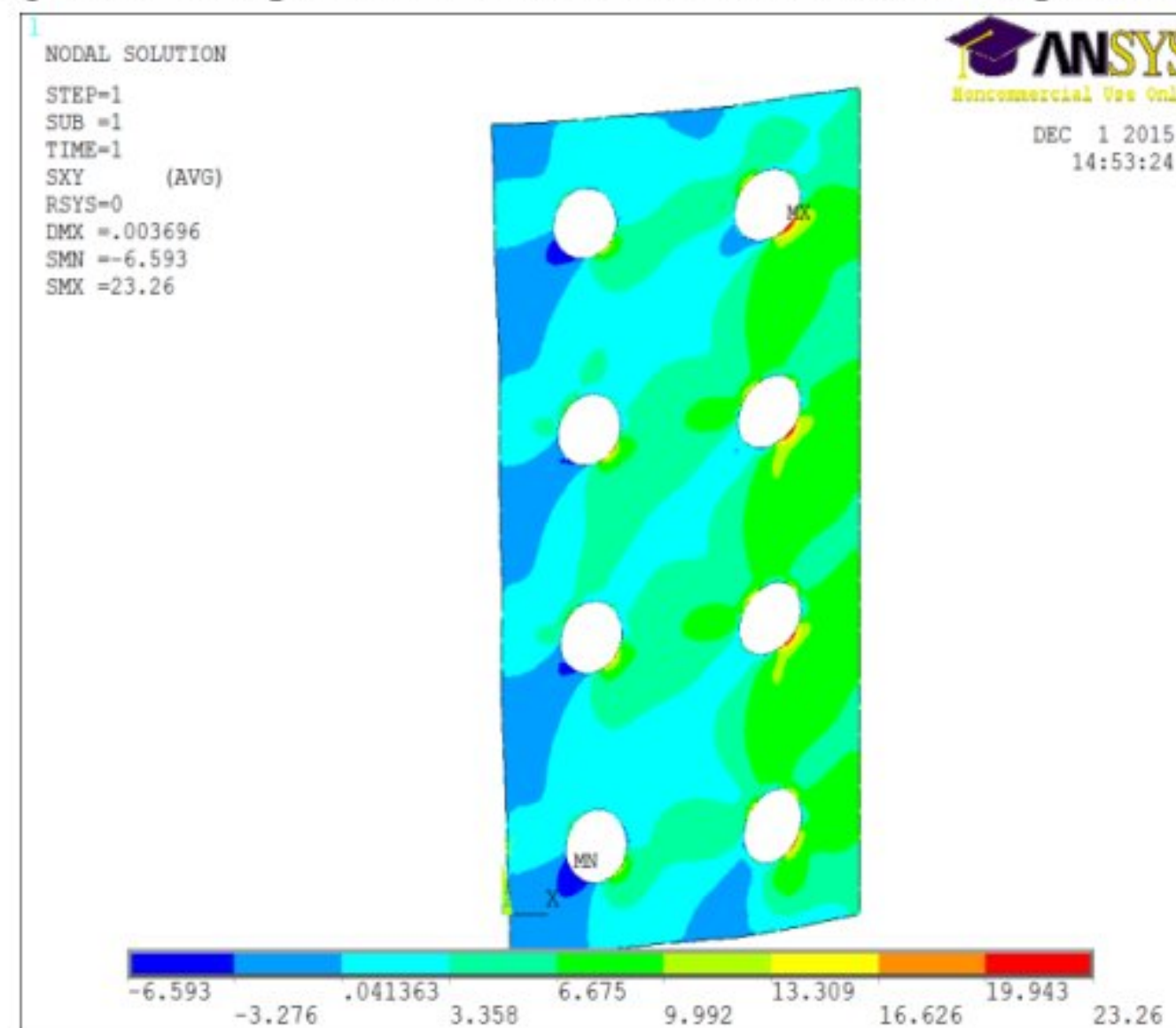
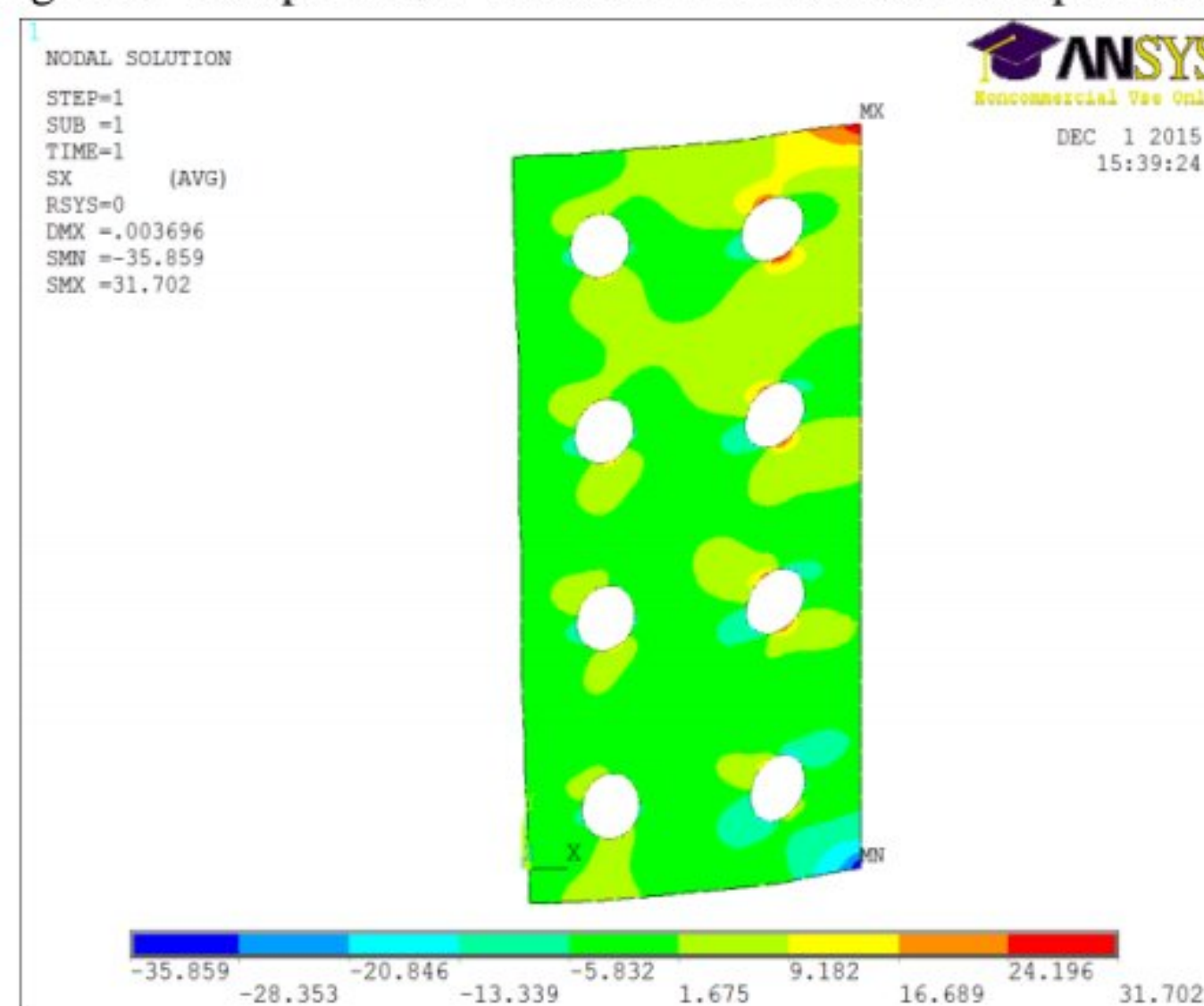


Figure 4 - Simple Plate - ANSYS Vertical Nodal Displacement



It can be seen in Figure 3 and Figure 4 that the stresses are accumulating in the correct locations. The maximum shear stresses are occurring in the middle region of the weld, and the maximum tension and compression stresses are occurring in the top and bottom corners of the weld respectively. However, the tension and compression stresses seem to have very large local build-ups

at the corners as opposed to being nicely distributed from positive to negative at the corners. This may cause the finite element analysis to result in much higher tension/compression stress than will be produced through standard elastic analyses.

6. Elastic Analyses

Shear Analysis

The maximum shear stress of a rectangular plate occurs at the mid-depth of the plate. It can be calculated by the following equation

$$\tau = \frac{3 \cdot V}{2 \cdot H \cdot (\text{in of thickness})}$$

As discussed earlier in the Final ANSYS Model section, the total force per bolt is 10 kips. With 8 bolts in the connection, the total force at the welded joint is 80 kips. With the total shear force at the connection ($V = 80$ kip) and the height of the plate ($H = 12$ in), the maximum stress at the middle of the plate can be determined as follows.

$$\tau = \frac{3(80 \text{ kip})}{2(12 \text{ in})(\text{in of thickness})}$$

This is almost identical to the shear stress of 9.992 ksi as determined by the ANSYS model. This proves the shear stress calculations used in practice are comparable to the finite element analysis results of a welded shear tab connection, at least for this geometry.

Compression/Tension Analysis

The geometry of this connection will result in a moment at the welded joint due to an eccentric loading. The tension and compression stresses at the extreme fibers of the plate due to moment effects can be determined using the following equation,

$$\sigma = \frac{M}{S}$$

Where M is the applied moment and S is the elastic section modulus of the shape. The resultant load from the 8 bolts occurs at the center of gravity of the bolt group. From Figure 1, it can

be determined that the resultant is located at an eccentricity, e , of 3 inches away from the welded joint. Therefore, the applied moment is calculated to be

$$M = V \cdot e = (80 \text{ kip})(3 \text{ in}) = 240 \text{ kip} - \text{in} = 10 \text{ ksi}$$

The section modulus of the connection plate is calculated as follows.

$$S = \frac{(\text{inches of thickness})(H)^2}{6} = \frac{(1 \text{ in})(12 \text{ in})^2}{6} = 24 \text{ in}^3$$

Now with all the required information, the stresses at the extreme fibers can be computed. In theory, the tension stress at the top of the joint should equal the compression stress at the bottom of the joint.

$$\sigma_t = \sigma_c = \frac{M}{S} = \frac{240 \text{ kip} - \text{in}}{24 \text{ in}^3} = 10 \text{ ksi}$$

These stresses are much lower than the tension and compression stresses determined in the finite element analysis.

7. Discussion of Results

Our objective was to compare the results of a finite element analysis of a welded shear tab connection to elastic analyses, or allowable stress design methods, used in structural engineering practice. The shear stress at the middle of the joint was computed to be 9.992 ksi using ANSYS, which was within 0.1% of the 10 ksi value calculated the using allowable stress design method. The maximum tension and compression stresses from the finite element analysis were far off. The compression stress computed by ANSYS was 35.9 ksi, which is over two and a half times larger than the 10 ksi value determined using the allowable stress design method. The Tension stress determined by ANSYS was also over two times larger than the 10ksi value determined with the allowable stress design method. The maximum tension stress computed by ANSYS was 31.7 ksi. As discussed earlier, it can be seen in Figure 4 that there are large stress build-ups at the ends of the welded joint. This could explain why the values for the tension and compression stresses computed by ANSYS were so much larger than those determined using the elastic analyses. The stresses are concentrating locally at the ends instead of distributing linearly from tension at the top to compression at the bottom as it does in the theoretical elastic equations. This is likely due to limitations of the configurations used in our ANSYS model. Also, the elastic analysis equations used in this project were based off Bernoulli-Euler beam theory which is not as applicable to short span beam action as other approaches such as Timoshenko Beam theory.

8. Conclusion

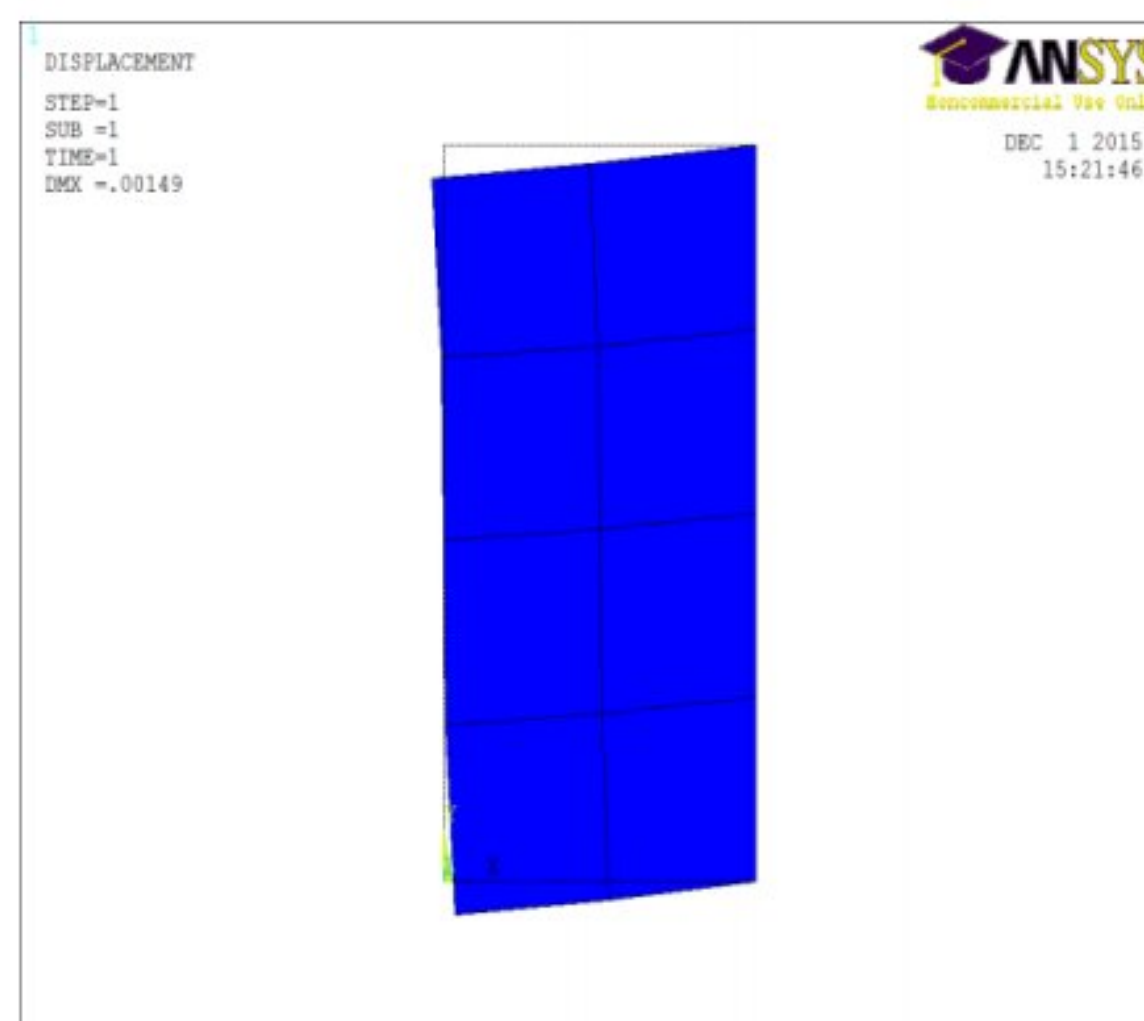
In conclusion, both the finite element analysis approach and the allowable stress design approach result in almost identical maximum shear stresses in the plate at the welded joint. The two approaches do not provide similar results for the maximum tension and compression stresses at the ends of the joint.

9. Appendix A – Simple Plate Model Convergence

Appendix A discusses the convergence of the model results for the simple plate arrangement used for the verification portion of this project. This model needed to converge when point loads were applied to all of the nodes in an effort to simulate an equivalent vertical area load on the face of the connection plate, as discussed in the Verification section. A point load of -1 was applied to each of the nodes for each mesh refinement. The result of interest was the vertical displacement of the plate at the free end. Because the number of nodes increased each re-meshing, the total load also increased. So to determine if the model was converging, we looked at the ratio of number of nodes to the vertical displacement at the end of the plate. The model converged to a 1% change in ratio after 4 meshes. See Table A below. The figures for the mesh and deformed shapes for each meshing can be seen in the following Figures A-1 through A-4.

Table A: Convergence of Simple Plate for Verification

Meshing	No. of Elements	Vertical Displacement	No. of Nodes	Node/Disp.	Percent Change
1	8	-1.49E-03	37	-24832	-
2	32	-4.61E-03	121	-26242	6%
3	72	-9.40E-03	253	-26918	3%
4	128	-1.59E-02	433	-27272	1%



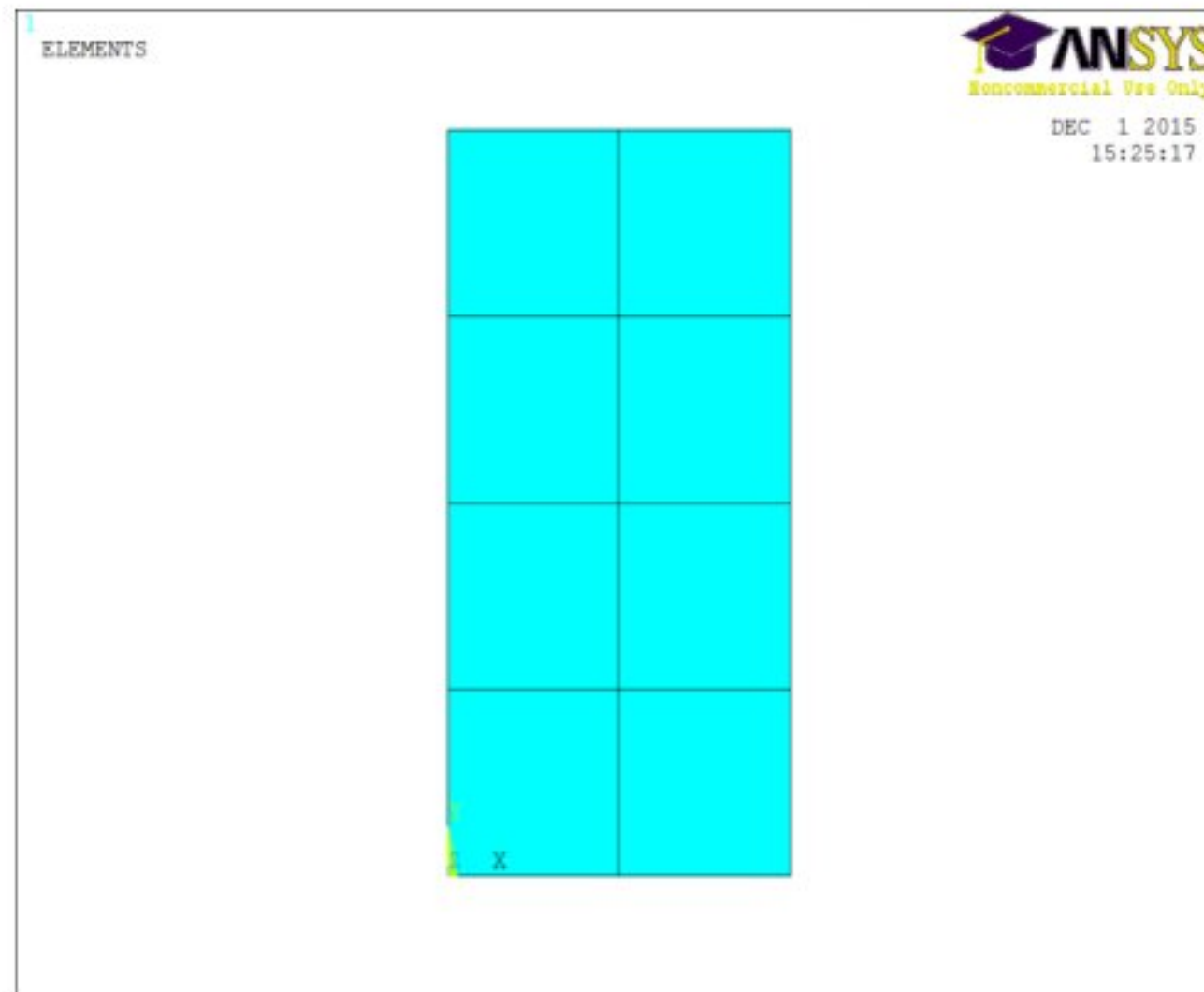


Figure A-1: Simple Plate - First Meshing

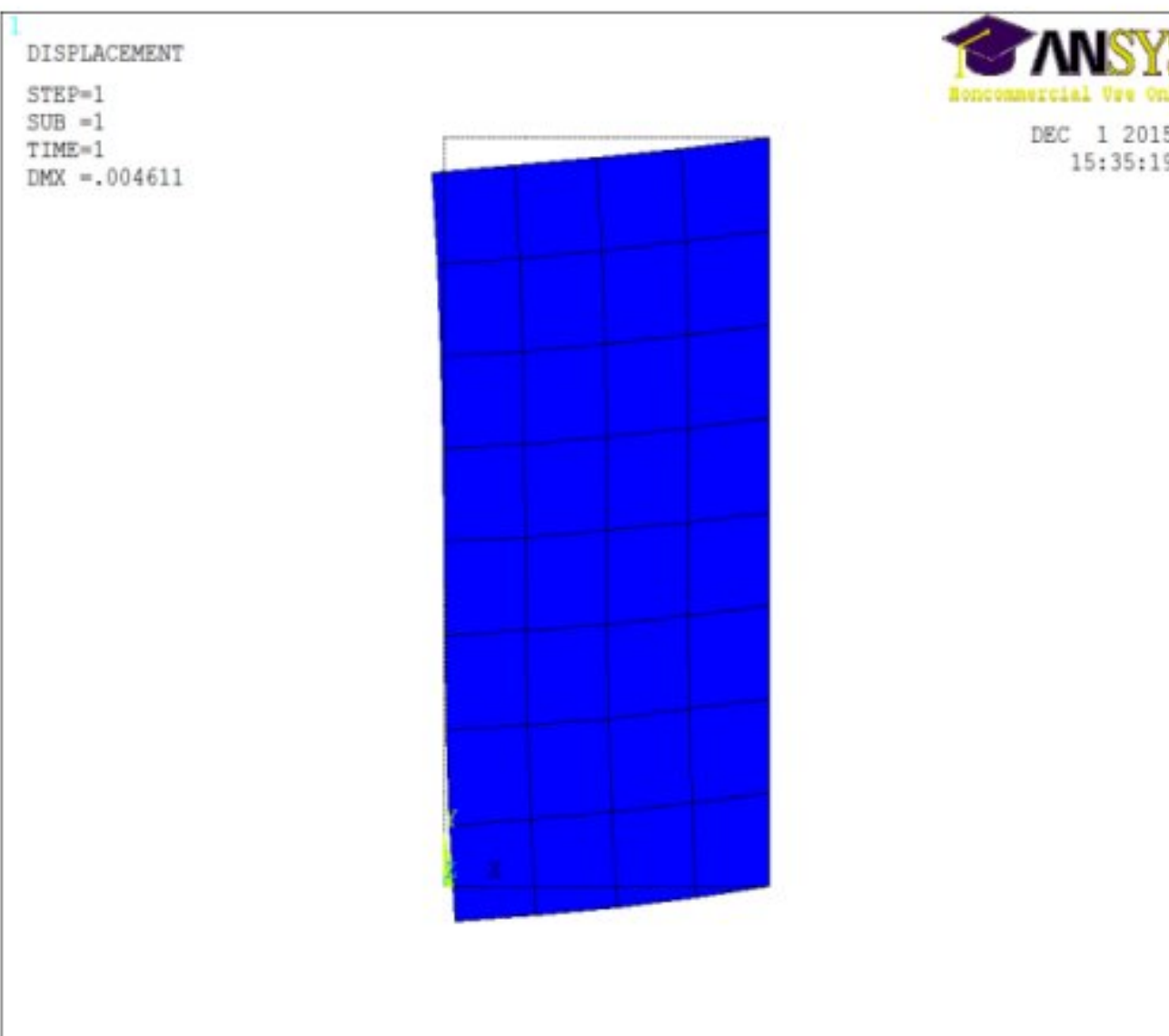
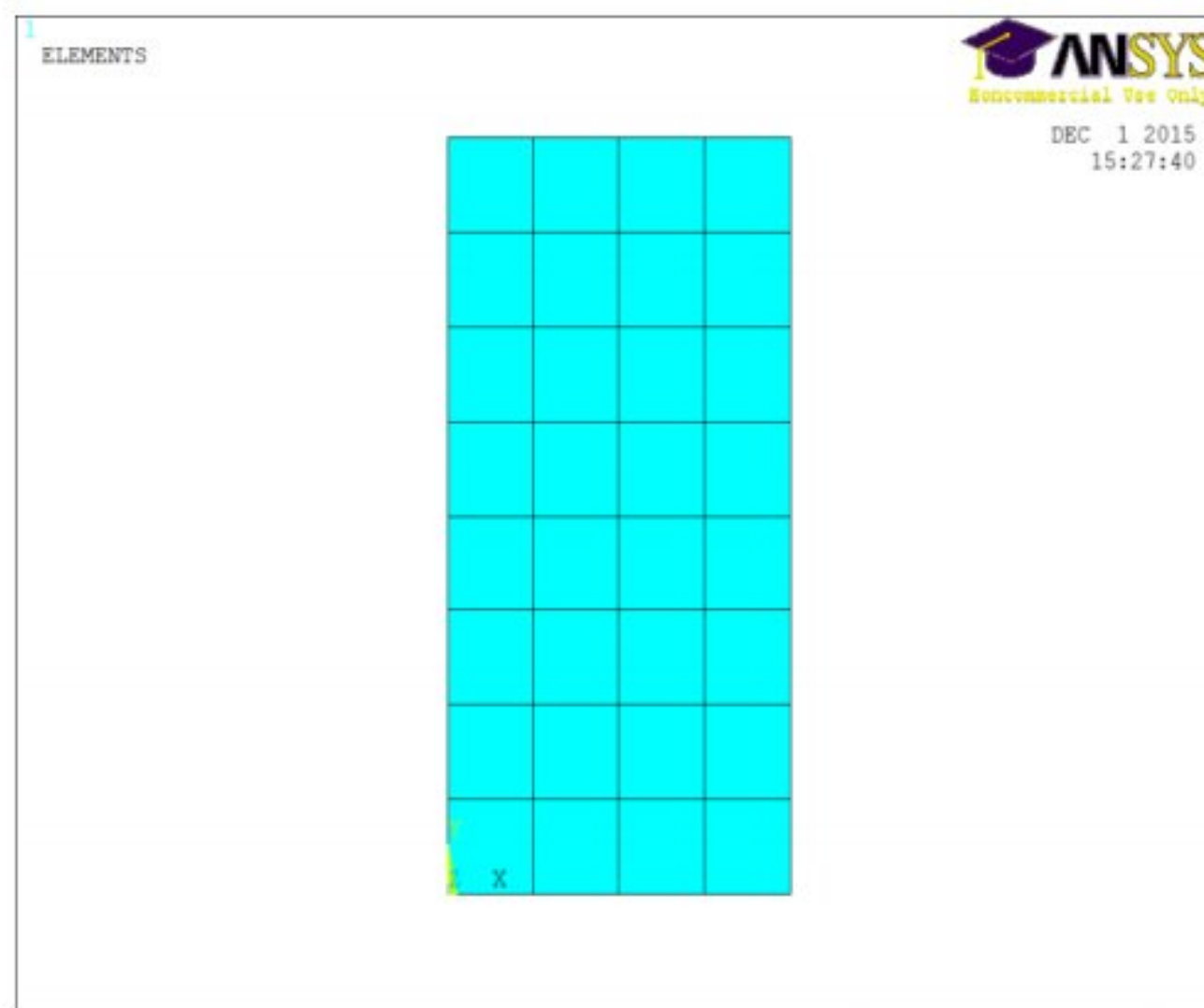


Figure A-2: Simple Model - Second Meshing

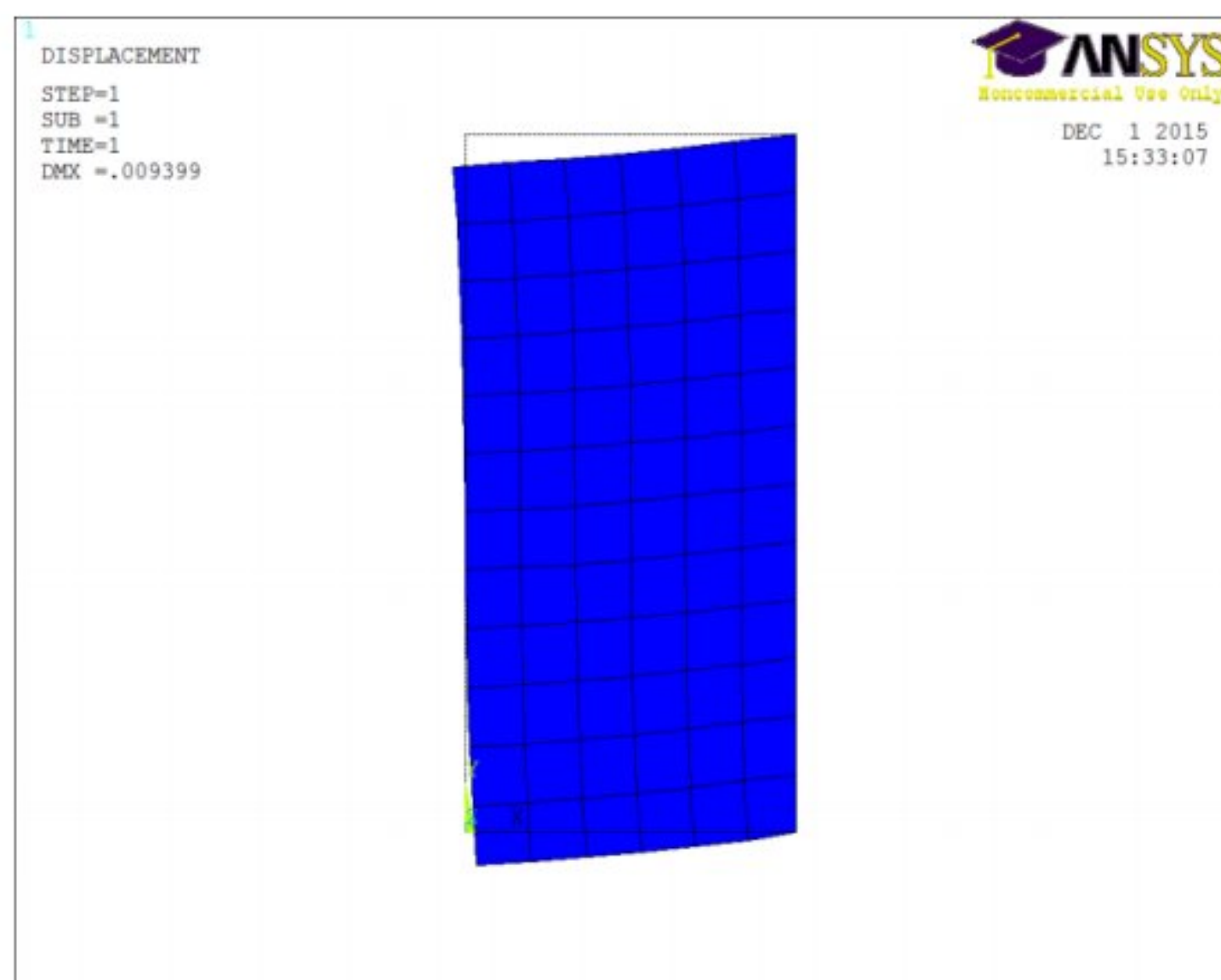
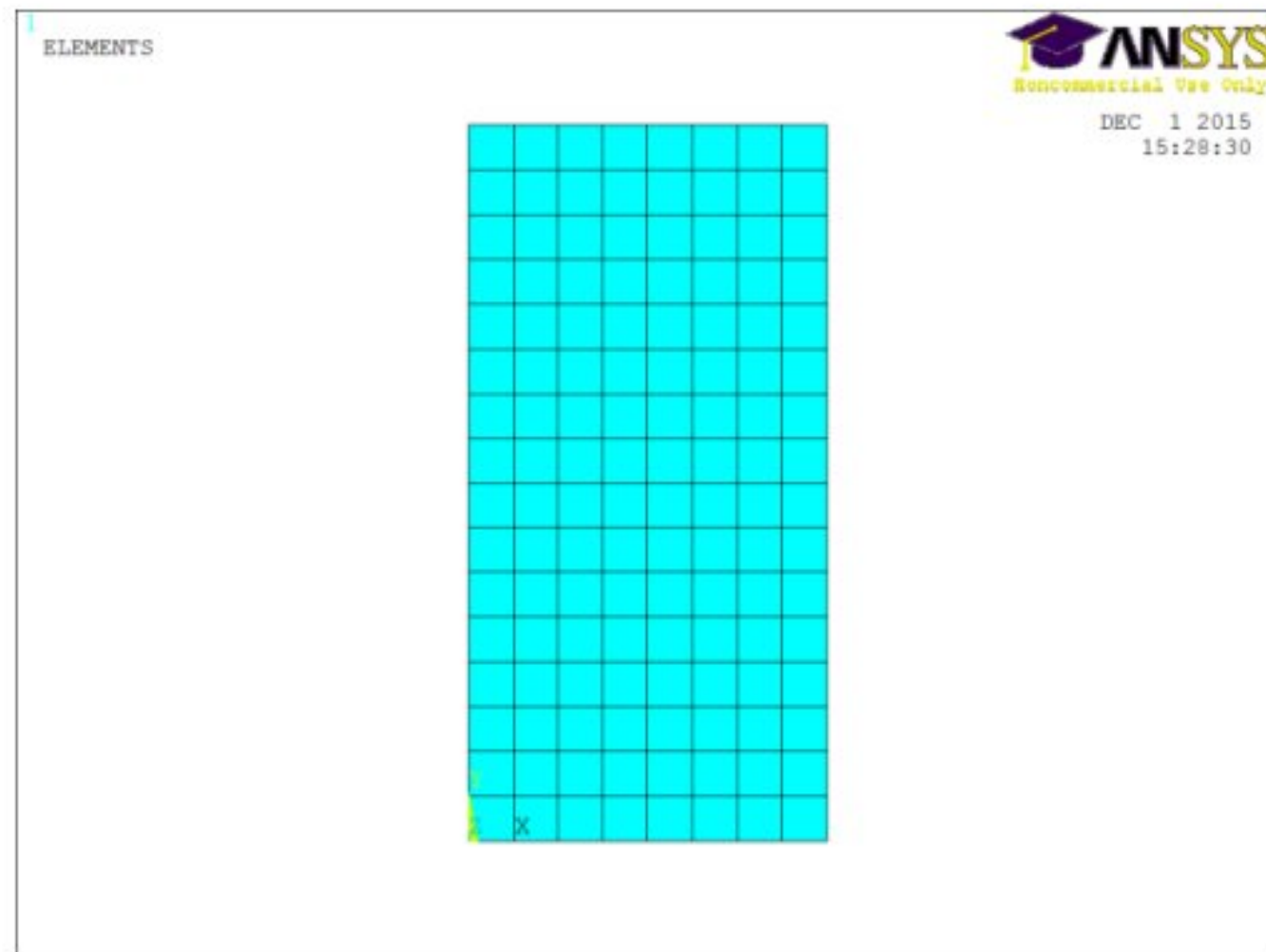
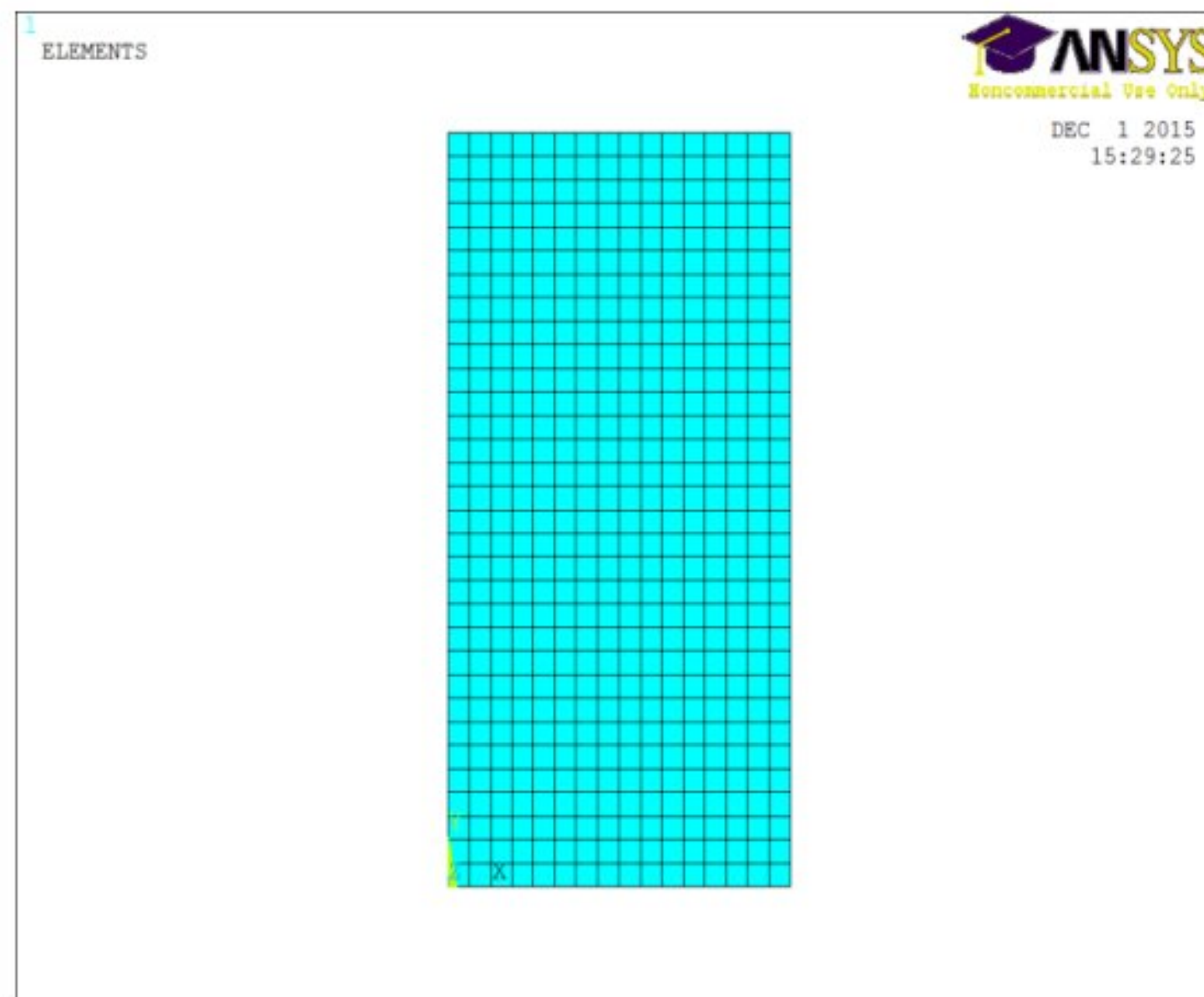


Figure A-3: Simple Model - Third Meshing



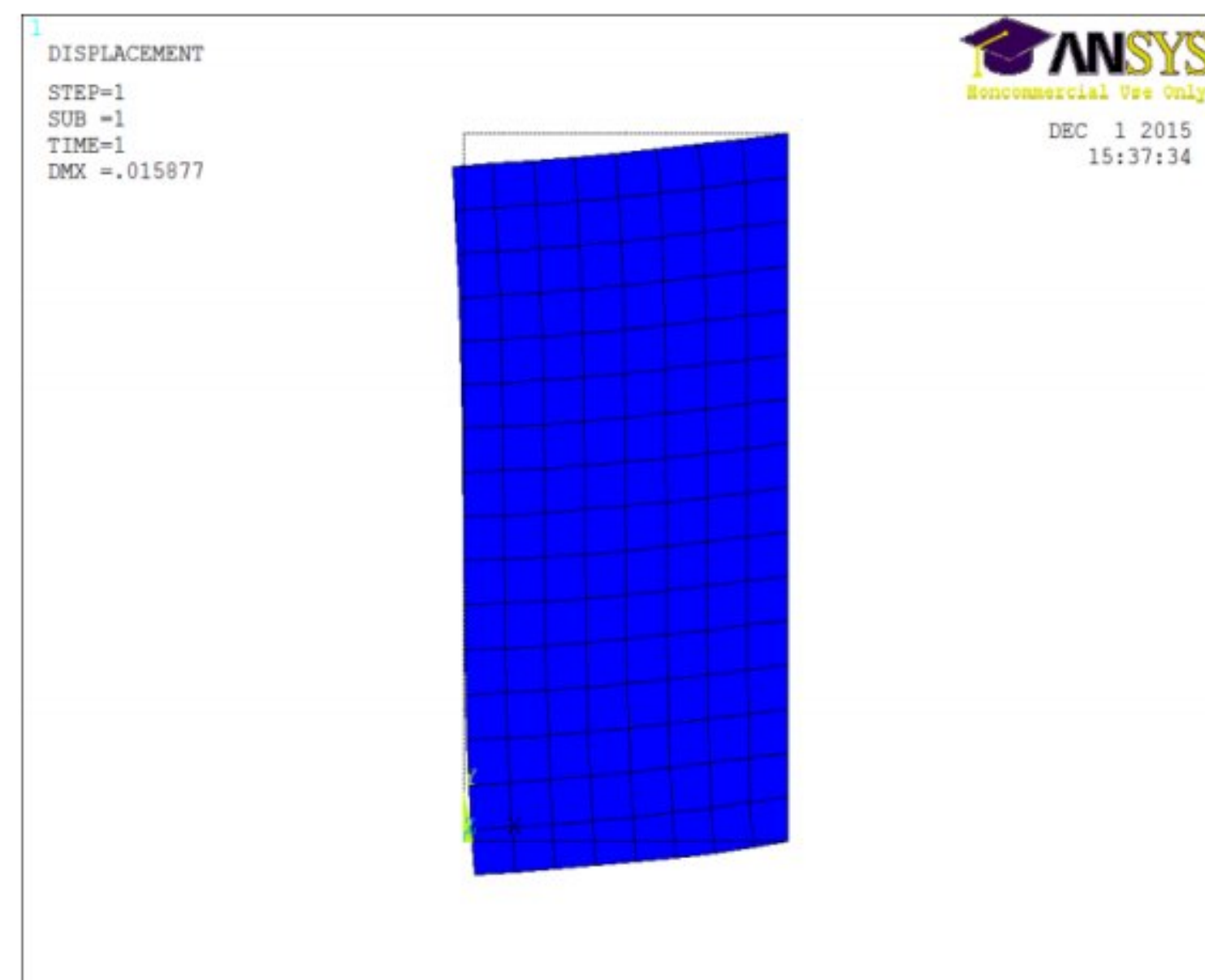
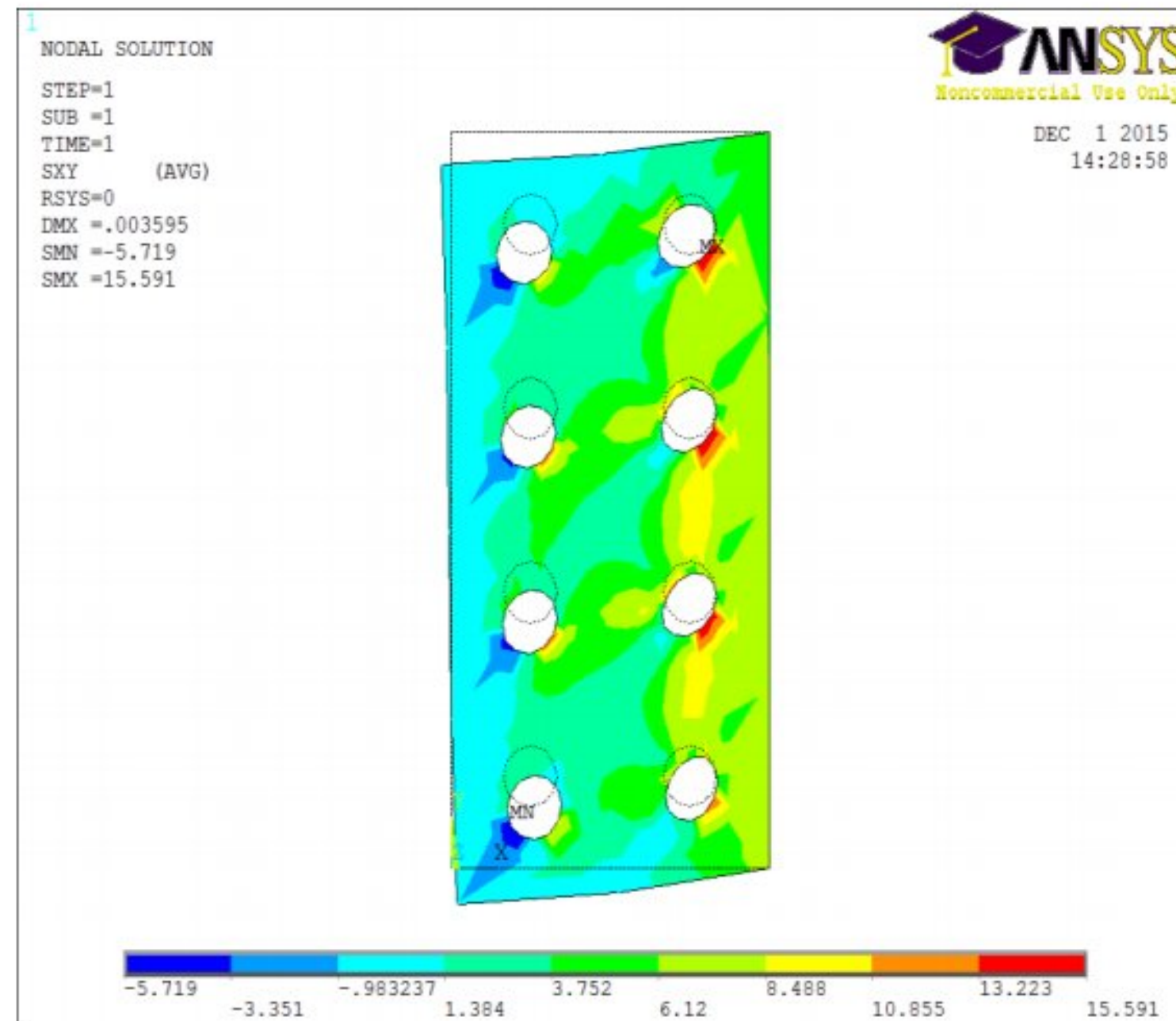


Figure A-4: Simple Model - Fourth Meshing

10. Appendix B – Final Model Convergence

Appendix B discusses the convergence of the shear stresses for the final model. We meshed this model until we reached the limit with this version of ANSYS. However, it can be seen that the model does converge by the final mesh. For this project, we are interested in the shear stresses near the middle of the joint. However, since the shears in that area See Table B below. The figures for the mesh and shear stress for each meshing can be seen in the following Figures B-1 through B-4.

Table B: Convergence of Final Model

Meshing	No. of Elements	Max Shear Stress	Percent Change
1	944	15.6	-
2	8496	21.3	36%
3	60416	23.3	10%
4	61696	23.3	0%

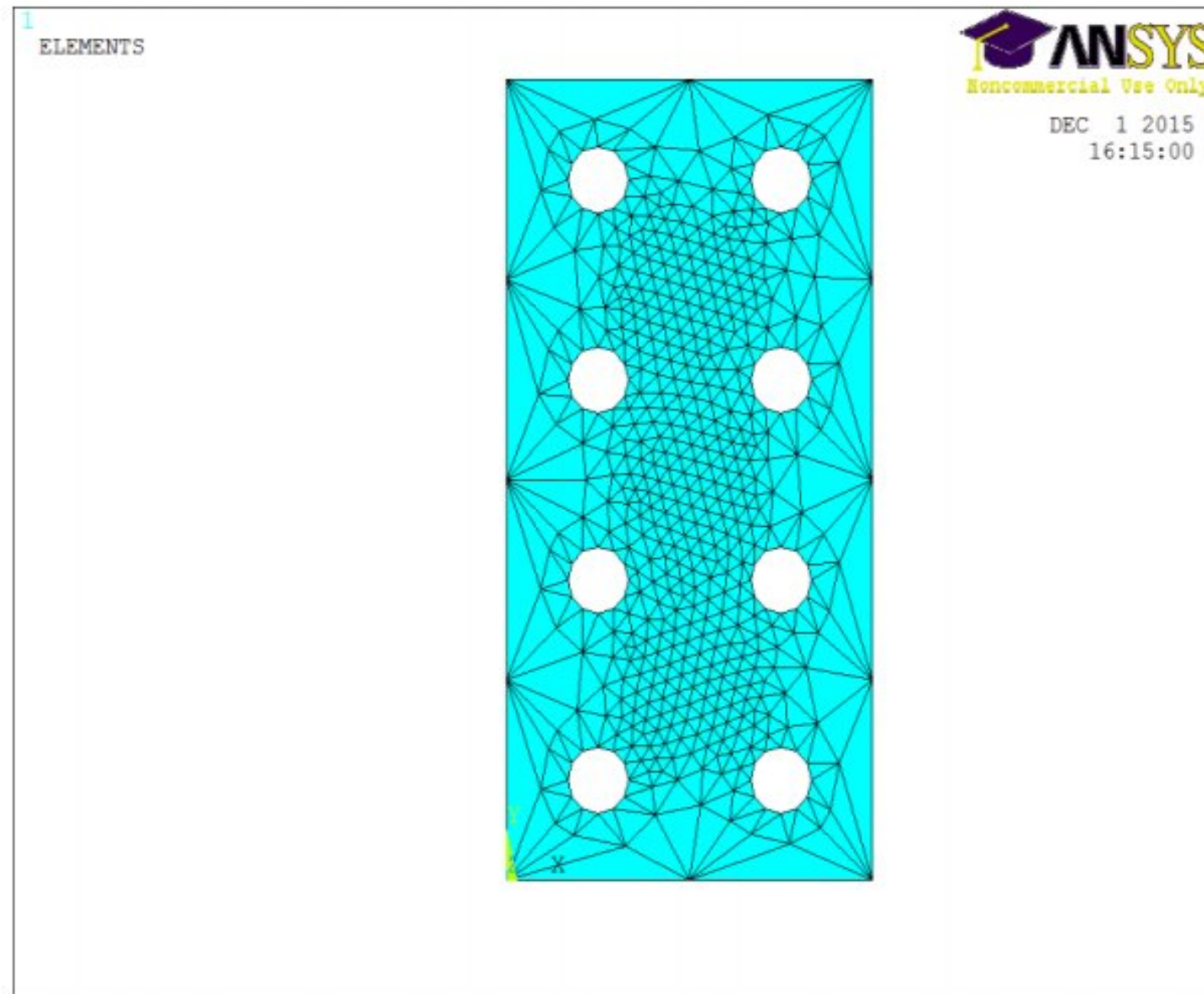


FIGURE B-1: Final Model - First Meshing



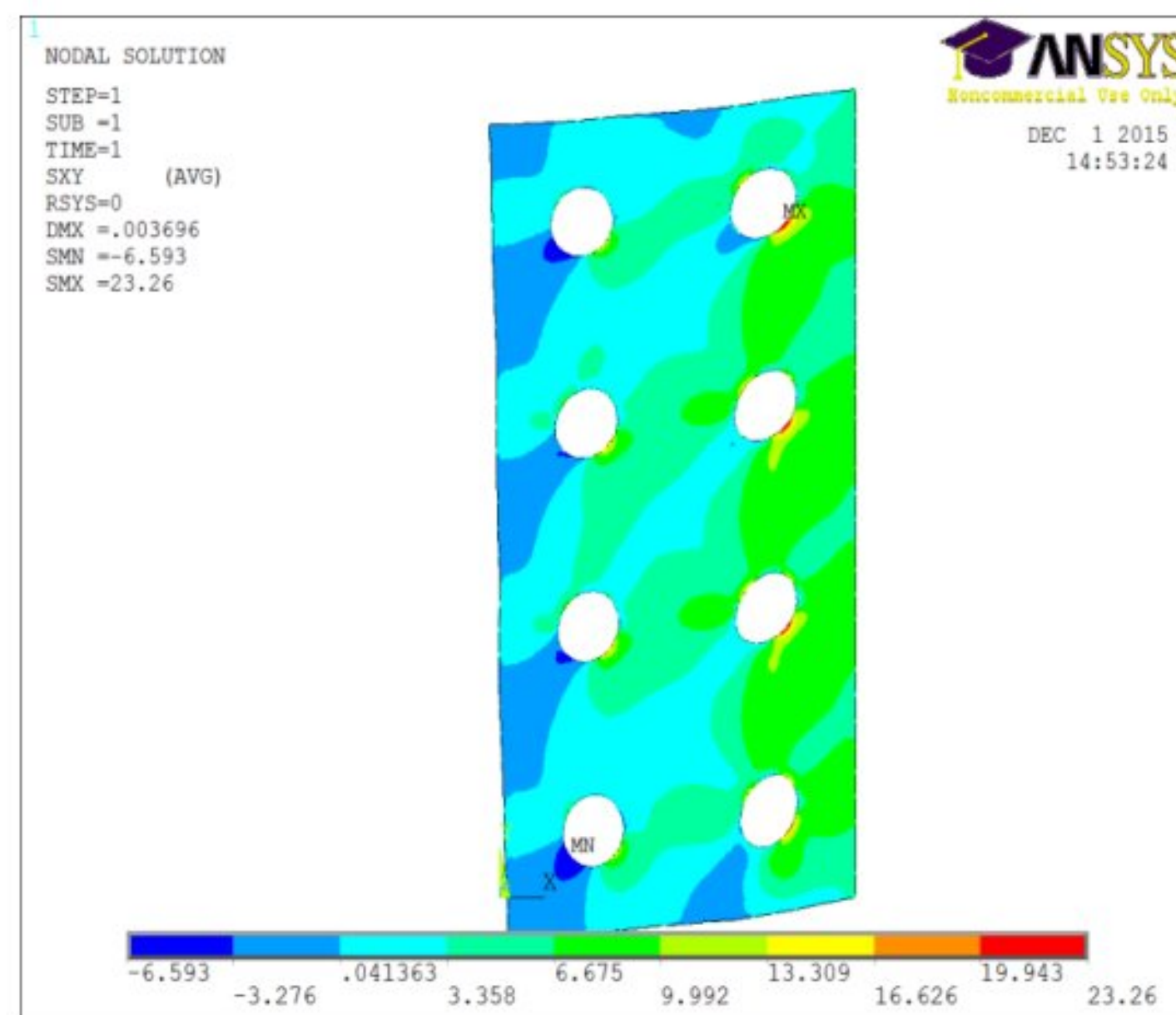
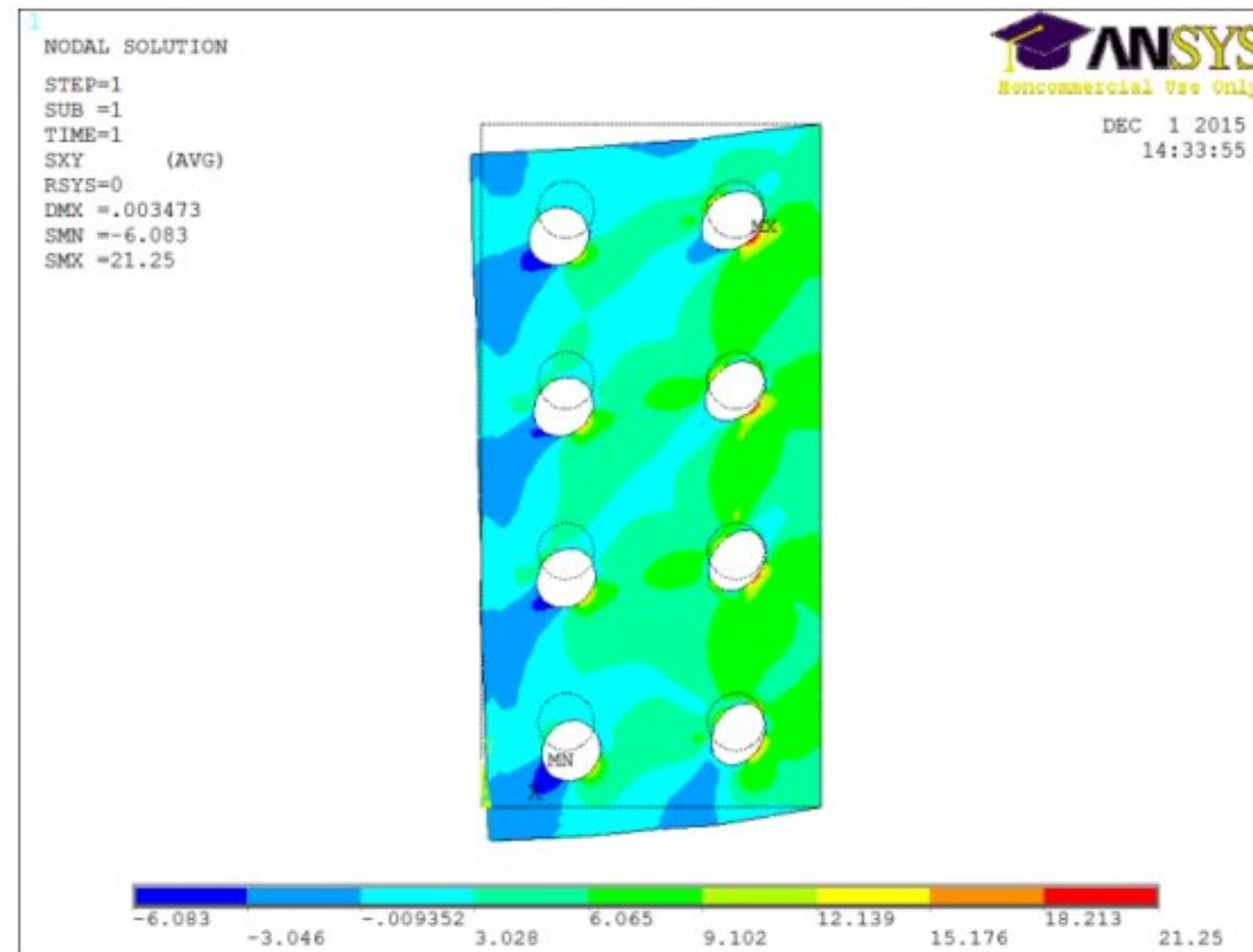
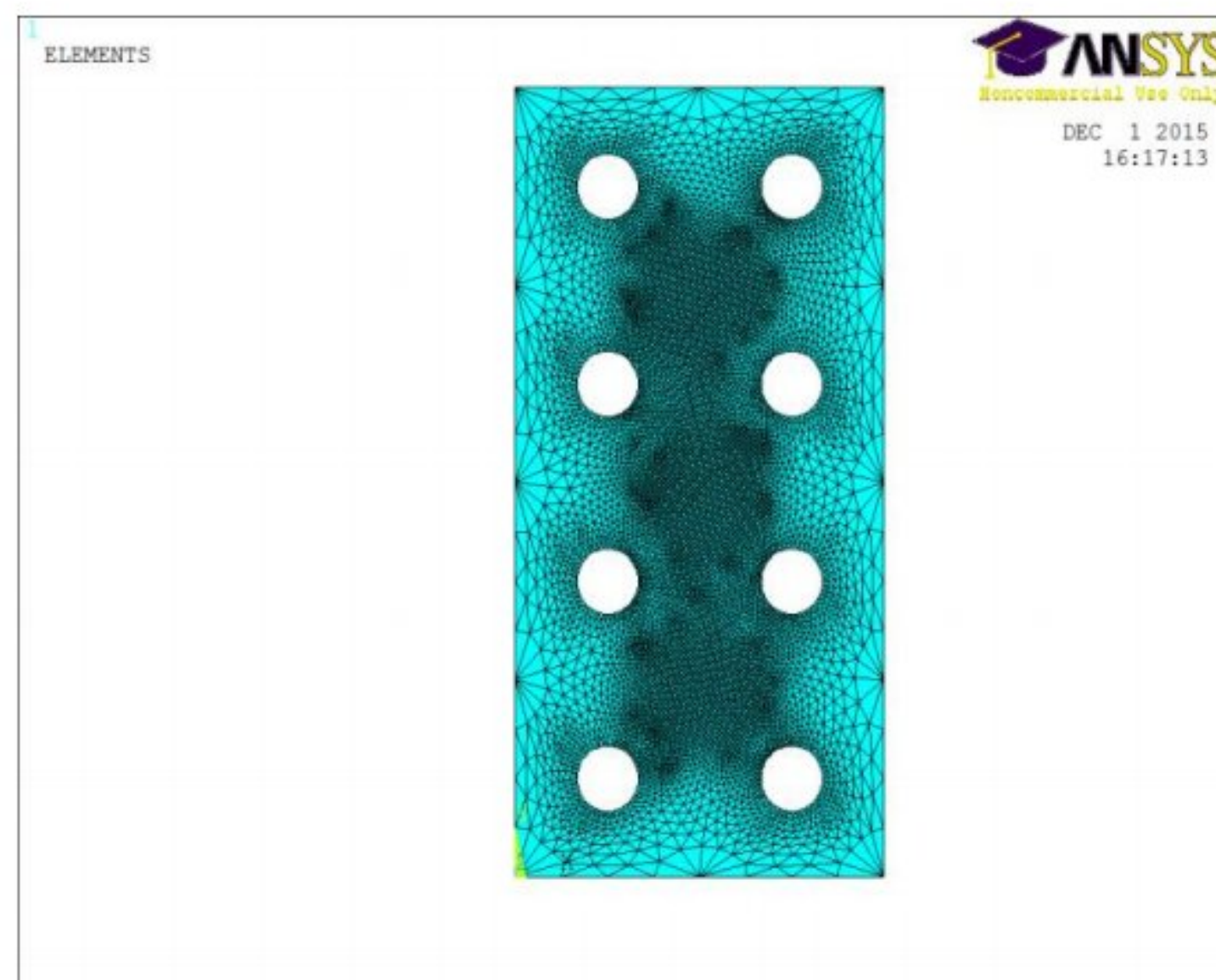


Figure B-2: Final Model - Second Meshing



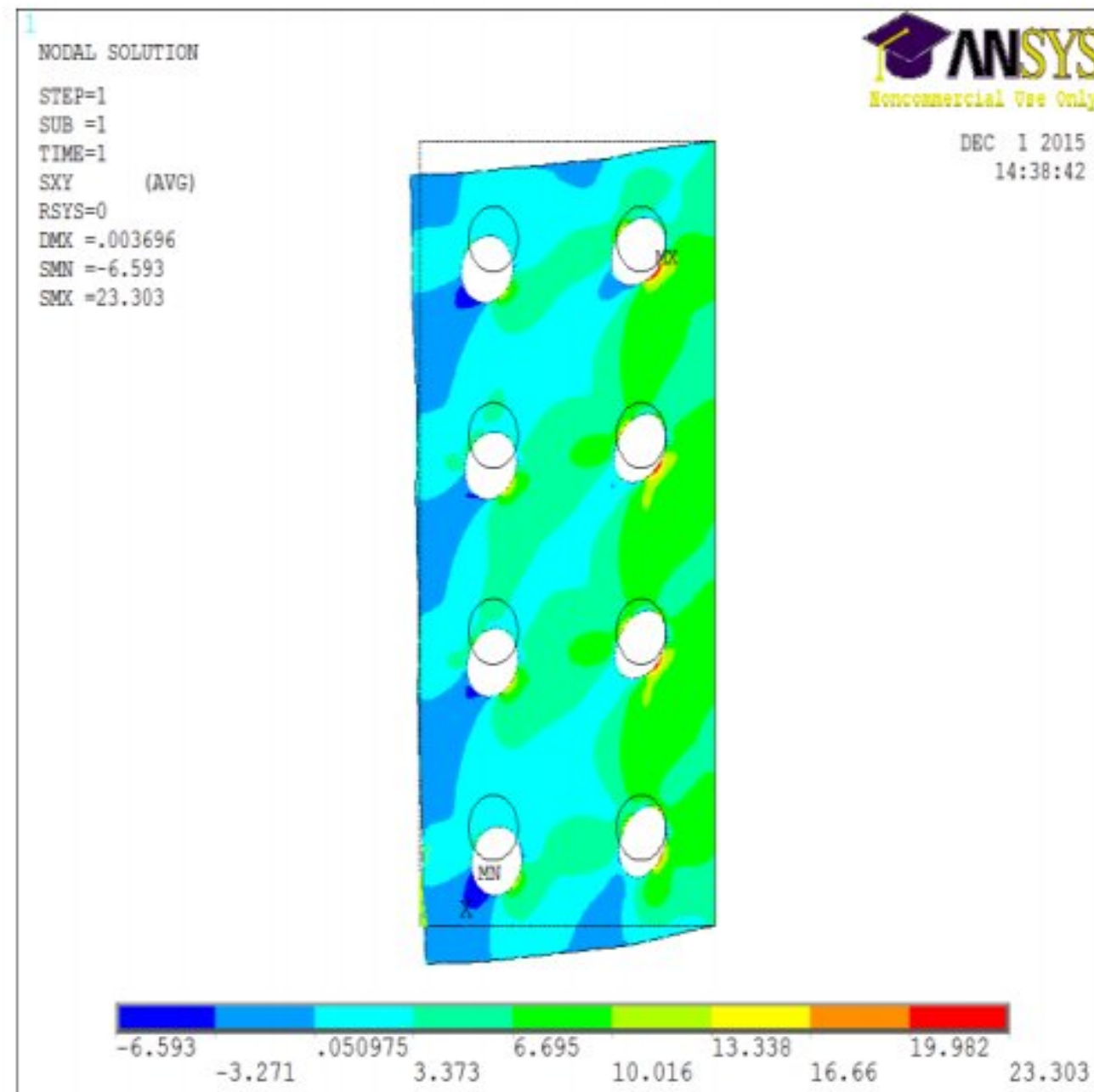


Figure B-3: Final Model - Third Meshing

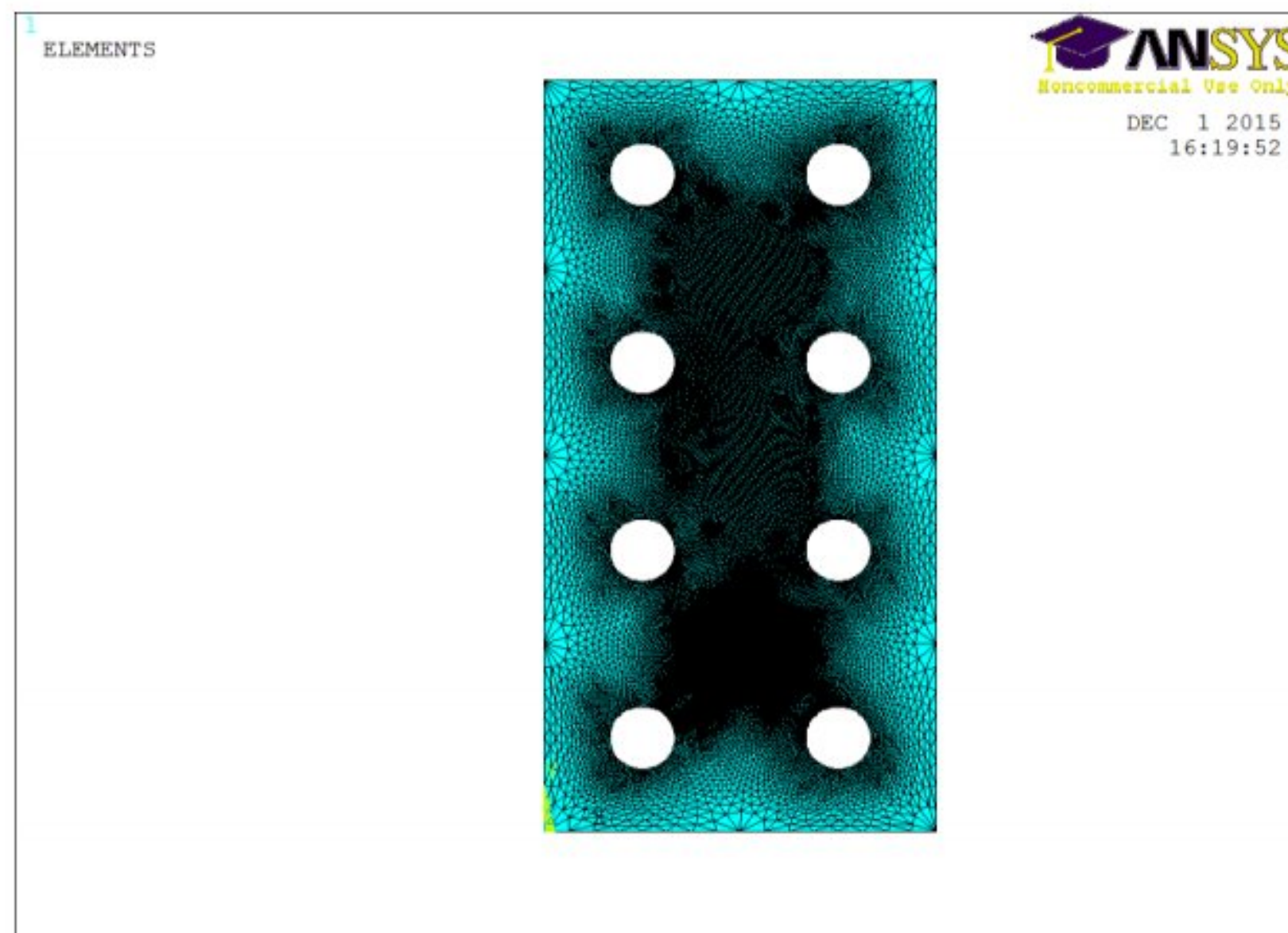


Figure B-4: Final Model - Fourth Meshing

Acknowledgment

I would like to thank Professor Craig D. Foster, Department of Civil and Materials Engineering, University of Illinois at Chicago and our, principal, Director, Management for their continuous encouragement and support in the utilization of latest software's from the college at Malla Reddy Engineering College (Autonomous), Secunderabad.

References

- Li Yajiang, Wang Juan, Chen Maoai and Shen Xiaoqin, 2004, Finite element analysis of residual stress in the welded zone of a high strength steel, *Bulletin of Material Science*, vol 27, No. 2, pp. 127-132.
- Wu A., Syngellakis S. and Mellor B.G., 2001, Finite element analysis of residual stresses in a butt-weld. *Proceedings of the Seventh Postgraduate conference in Engineering Materials*, 37-38
- Goldak J., Chakravarti A., and Bibby M., 1984, A New Finite Element Model for Welding Heat Sources, *Metallurgical Transactions B*, 15B: pp.299-305.
- Peng-Hsiang Chang, Tso-Liang Teng, 2004, Numerical and experimental investigations on the residual stresses of the butt-welded joints, *Computational Materials Science*, Vol 29, pp.511–522.
- Tsirkas S.A., Papanikos P., Kermanidis Th., 2003, Numerical simulation of the laser welding process in butt-joint specimens, *Journal of Materials Processing Technology*, Vol 134, pp.59-69
- Camilleri D., Mollicone P. and Gray T G F, 2006 Alternative simulation techniques for distortion of thin plate due to fillet-welded stiffeners, *Modelling and Simulation in Material science And engineering*, Vol 14, pp.1307-1327

Author Profile



Er. B. Vamsi Krishna, M.I.S.T.E, M.I.E, C. Engg., did his Bachelors in Civil Engineering and also obtained Masters Degree in Structural Engineering from University of Salford, Manchester, UK. He worked as Junior Bridge Engineer in ERA Infra Projects Ltd and also as contractor in L&T Metro Rail, Hyderabad and is currently working as Assistant Professor in the Department of Civil Engineering, Malla Reddy Engineering College (Autonomous). He has very good Industrial knowledge and also extensive teaching skills in Concrete Technology, Analysis and Designs of Structural Engineering, Strength of Materials e.t.c. He has published 18 research papers at International Journals, 8 research papers in International & National Conferences and 2 Magazines. His area of research interest includes Health Monitoring of Structures, Advances in Concrete Technology and Design of RCC & Steel Structures.