# Abstract: report to analyse a truss frame

Abstract: This report is set tocompare finiteelement analysis with a theoretical model to verify its validity. DassaultSystèmes' Abaqus is utilized in this report to analyse a truss frame model. Thefindings of both models are viable and the differences can be ignored due to accuracy; theoretical model yields values with low number of significant figures. Abbreviations: FEA, Finite element analysis. 1.

0 Introduction: Engineering components can be of diverse shapes, intwo1or three2dimensions. In this report, we confined our discussion to linearelastic analysis. The success rate of the numerical analysis is dependent onthe comparison with the theoretical method. FEA aids in understanding thebehaviour of constrained models, to extrapolate the performance; by tocalculating the safety margin. To amend model to optimal design. The precisionof numerical model highly depends on the approximation of all initial parameters. 2.

0 TheoreticalModel: This model is to be used in comparison with thenumerical model. This model fails to take variables like young modulus, Eor passion ratio into account. This is predicted to show a minor significance inless accurate notations. However, higher accuracy yields differences in lowerprefixes.

2. 1 Support Reactions: Calculations for the fixedpin at node 1: Calculations for rollerpin at node 2: Where: A is the reactionforce at the pin and all angles are in degrees. 2. 2 Forces in all elements: At node 4: At node 3: 3. 0 Numerical Model: Finite element analysis, or FEA, is an approximate method of solving engineering problems. Generallyused to solve problems with no

exact solution. It’s rather anumerical method than analytical. The need to FEA arises as analytical methodscannot handle complex problems like engineering problems.

For instance, we cancalculate, analytically, the stresses and strains in a bent beam usingengineering mechanics of materials or the mathematical theory of elasticity. But neither will be very effective in determining what is happening in part ofa car suspension system in cornering. The first appearance of FEA originated tofind stresses and strains in engineering models under concentrated loads. FEAdemands a large amount of computation power.” The method is now applied to problems involving awide range of phenomena, including vibrations, heat conduction, fluid mechanicsand electrostatics, and a wide range of material properties, such aslinear-elastic (Hookean) behaviour and behaviour involving deviation fromHooke’s law (for example, plasticity or rubber-elasticity).

” says Modlen, G.(2008). Figure 3. 0. 1.

Initial Model, in white. Model, rainbow, under applied force at node 4Shown in figure 3. 0. 1 is the resultant concentratedforce applied to node 4.

The white line represents the frame when no forcesapplied. The frame has a Young’s Modulus, E of 210 GPa. Poisson’s ratio3, at0. 31. Geometrically, the frame has 4 nodes and 5 elements.

An element length of1m and a circular profile diameter of 15 mm. It’s a frame; since it’s equilateral sinceall elements are equal. This also means angles are 60º degrees. Node 1 is afixed pin and node 2 is a roller.  The magnitude of  equals to 2kN as shown in the figure 3. 0.

1. Asa result of the applied force, shown in the contoured graph, the frame is seento deform and change in mechanical and geometrical values. 3. 1 Nodal DisplacementsThe displacement of a respective point withinan element is fixed by the displacements of nodes of an element, that is afunction of the nodal displacements. The system matrix is simply asuperposition of the individual element stiffness matrices with properassignment of element nodal displacements and associated stiffness coefficientsto system nodal displacements. Figure 3.

1. 1 presentsthe magnitudes of displacement, U in nodes3. 2. 1 Nodal ForcesTable 3. 2.

1. 1 presentsthe active nodal forces     Node Label    RF. Magnitude          RF. RF1          RF. RF2            U. U2         S.

Mises                          @Loc 1         @Loc 1         @Loc 1         @Loc 1         @Loc 1

——————————————————————————————————————————-

1          3. E+03    -113. 687E-15          3.

E+03    -3. 00000E-33     7. 62394E+06              2     1. 00000E+03     0.    -1.

00000E+03     1. 00000E-33     4. 90111E+06              3              0.     0.              0.     4. 49152E-06     6. 53481E+06              4              0.

0. 0. -130.

254E-06 9. 80221E+06 Minimum 0. -113.

687E-15 -1. 00000E+03 -130. 254E-06 4. 90111E+06 At

Node 4 1 2 4 2

Maximum 3. E+03 0. 3. E+03 4.

49152E-06 9. 80221E+06 At Node 1 4

1 3 4 Total 4. 00000E+03 -113. 687E-15 2.

00000E+03 -125. 762E-06 28. 8621E+063. 2.

2 SupportReactionsTable 3. 2. 2. 1 presentsthe acting forces on elements

Part Instance Node ID Attached elements RF, Magnitude

——————————————————————————— TRUSS-1 1 1

3. E+03 TRUSS-1 1 4 3. E+03 TRUSS-1

1 5 3. E+03 TRUSS-1 2 1 1. E+03

TRUSS-1 2 2 1.

E+03 TRUSS-1 4 3 0. TRUSS-1 4

4 0. 3.

3 Forces in allelementsOutput sorted bycolumn “ Element Label”. Averaged

at nodesTable 3. 3. 1 presentsthe acting forces on elementsNode Label RF.

Magnitude RF. RF1 RF. RF2 U.

U21 @Loc 1 @Loc 1 @Loc 1 @Loc 1

———————————————————————————————————————-1 3.

E+03 -113. 687E-15 3. E+03 -3.

00000E-33 7. 62394E+062 1. 00000E+03 0. -1.

00000E+03 1. 00000E-33 4. 90111E+063 0. 0.

0. 4. 49152E-06 6.

53481E+064 0. 0. 0. -130. 254E-06 9.

80221E+06Minimum 0.

-113. 687E-15 -1. 00000E+03 -130.

254E-06 4. 90111E+06At Node 4 1 2

4 2Maximum 3. E+03 0. 3.

E+03 4. 49152E-06 9. 80221E+06At Node 1

4 1 3 4Total 4. 00000E+03 -113.

687E-15 2. 00000E+03 -125. 762E-06 28.

8621E+063. 4 Stress in allelementsStress analysis for trusses, beams, and other simple structures arecarried out based on dramatic simplification and idealization: – massconcentrated at the center of gravity – beam simplified as a line segment (samecross-section) • Design is based on the calculation results of the idealizedstructure & a large safety factor (1. 5-3) given by experience. Table 3.

4. 1 presentsthe stress in elements PartInstance Element ID Type Int. Pt. S, Mises

——————————————————————————————– TRUSS-1 4 T2D2 1 13. 0696E+06 TRUSS-1 3 T2D2 1 6. 53481E+06 TRUSS-1 5 T2D2 1 6.

53481E+06 TRUSS-1 1 T2D2 1 3.

2674E+06 TRUSS-1 2 T2D2 1 6. 53481E+064. 0

Analysis & Discussion: 4. 1 Material discernment: Poisson’s’s Ratio, Elastic Moduli & Mohs ScalePoisson’s’s ratio, ? is the ratio of transversecontraction strain to longitudinal extension strain in the direction ofstretching force. In comparing a material’sresistance to distort under mechanical load rather than to alter in volume, Poisson’s’sratio offers the fundamental metric by which to compare the performance of anymaterial when strained elastically. Molybdenum, as shown in figures, has a toughness of 5. 5— bypassing iron.

Allowing it to beused in many industrial applications. A young modulus of 4. 7700E4 Pa, making itelastically suitable for a truss frame. Has an atom around twice as heavy as aniron atom, which allows for better crystallization processes. 4. 2 DiscussionFEA model values are instant values of steps; thehigher the number, the more the model can be manipulated.

The calculated theoreticalmodel failed to include the elastic moduli, as well as Poisson’s’s ratio. It isshown in some results, i. e. nodal force number 1. 5. 0 Conclusions: In finite elementanalysis, solution accuracy is judged in terms of convergence as the element” mesh” is refined. FEA models, when defined correctly, are more representativeof real life emulation. In FEA the thickness of the elements was 1.

767E4. Whereas the theoretical model failed to include the thickness. Both models are viableand the differences can be ignored due to accuracy; theoretical model yieldsvalues with a low number of significant figures. For purposes of comparison, analytical model valuesare to be rounded, resulting in matching values. Bibliography: Widas, P.(1997) ‘ Introduction to Finite Element Analysis’. Virginia Tech MaterialScience and Engineering online (January 2008), 87.

available from Greaves, G.

N., Greer, A. L., Lakes, R. S., and Rouxel, T.

(2011) ‘ Poisson’s’s Ratio andModern Materials’. Nature Materials 10 (11), 823–837Gray, T., Mann, N., and Whitby, M. (n. d.) The Photographic Periodic Table of theElements online available from

com/> 13December 2017Tura, A., Dong, Z., Trivedi, S., Shibata, Y., Tanimoto, Y.

, Maruyama, N., Nagakura, M., Roylance, D., Pérez-Pevida, E., Brizuela-Velasco, A.

, Chávarri-Prado, D., Jiménez-Garrudo, A., Sánchez-Lasheras, F., Solaberrieta-Méndez, E., Diéguez-Pereira, M., Fernández-González, F.

J., Dehesa-Ibarra, B., Monticelli, F., Parashar, S. K.

, Sharma, J. K., Mohammed, S., Desai, H.

, Marci, P., Florian, Z., Bart, S., Luan, Y., Guan, Z., Cheng, G., Li, W.

, Rungsiyakull, C., Field, C., Lin, D., Zhang, L., Li, Q., Swain, M., Gröning, F.

, Fagan, M., O'Higgins, P., Evans, S.

P., Parr, W. C. H., Clausen, P. D., Jones, A.

, Wroe, S., Du, J., Lee, J. H., Jang, A.

T., Gu, A., Hossaini-Zadeh, M., Prevost, R., Curtis, D. A., Ho, S. P.

, Dalla Pria Bankoff, A., Choi, A. H., Conway, R. C., Taraschi, V.

, and Ben-Nissan, B. (2012) ‘ A Review of Improved Fixation Methodsfor Dental Implants. Part II: Biomechanical Integrity at Bone-ImplantInterface’. Journal of Biomechanics online 2016 (5), 2702–2705. available from 1 Membrane, plate, shell, etc…2 3D fields can be: temperature, displacement, stress, flow velocity, etc…1 This report was submitted as for the completionrequirements of 208MAE/CWK1—practical finite element analysis. 3 Refer to chapter 4. 1for a point of reference.