Journal of Basic and Applied Engineering Research
p-ISSN: 2350-0077; e-ISSN: 2350-0255; Volume 3, Issue 3; January-March, 2016, pp. 218-222

http://www.krishisanskriti.org/Publication.html

Workability Analysis of Steel using Finite Element Method

Jibin Babu[1], Akhilchandran B S[2] and Akhil C.[3]

[1,2,3]VIT University, Vellore
E-mail: [1]jibinbabu28@gmail.com, [2]akhilchandranbs@gmail.com, [3]cakhil31@gmail.com

Abstract—*Workability of a material is defined as its capability to be put into efficacious operations, in other words how efficient the material is to be dealt with, worked on or to be handled. Steel is one of the most consequential alloys being used from antediluvian times. Alloy steel 42CrMo4 is an excellent material for gears, piston pins, crankshafts etc. This steel is widely utilized in a variety of applications in the oil and gas industry. The paper deals with the workability study of steel under sundry working conditions utilizing FEM software Autodesk Inventor. FEM, or Finite Element Method, is a mathematical technique used to soothsay the replication of structures and materials to environmental factors.Its a puissant engineering implement, to numerically simulate the authentic world without the desideratum to test prototypes in a lab. Steel specimens of cylindrical and tapered cross section are modeled. Steady load of 1 tonn is made to act on the top surface of the specimens under prescribed boundary conditions. The goemetrical transformation of the specimen is studied. The stress and strain values due to load applications are additionally evaluated. The deformed cylinrical and tapered steel models are compared and the results are interpreted.*

Keywords*: Steel, 42CrMo4, cylinrical, tapered, Autodesk Inventor*

1. INTRODUCTION

Workability of a material is defined as the degree of ease by which a material can be cut, shaped, or smoothed by hand or machine. Workability of a material may depend upon sundry factors like geometry of the material, lubrication conditions, stress- strain conditions, temperature etc. The study of workability has been in great increse to identify the capability of the material to act and respond at sundry working conditions and transmuting environment. **42CrMo4 Steel is a prevalent Chromium-Molybdenum alloy steel that conventionally used after quenched and tempered. DIN 1.7225 has better performance than 1.7220 steel due to the carbon and chromium content is higher. 42CrMo4 steel has higher vigor and hardenability. This material additionally has high fatigue vigor and good low-temperature impact toughness. The temper brittleness is not obvious.**

Table 1.1: Chemical composition

C	Si	Mn	P	S	Cr	Mo
0.38-0.45	Max 0.4	0.6-0.9	Max 0.026	Max 0.035	0.9-1.2	0.15-0.3

Its very prevalent Quenched & Tempered alloyed engineering steel grade with high intensity, high hardenability. This steel is generally utilized in the manufacture of implements which has high vigor requisites. This steel is good for immensely colossal size and paramount components, such as axle, gear, connecting rod, gear transmission, turbocharger gear, engine cylinder, spring, oil drill pipe joints and fishing implement. 42CrMo4 steel is sometimes used in lieu of the high nickel quenched and tempered steels.

Table 2: Mechanical properties

Tensile strength	1080MPa
Yield point	930MPa
Percentage reduction of area after fracture	45%
Elongation after fracture	12%
Impact test (+20°C)	Transverse–22J Longitudinal – 33J

It’s mainly utilized for making sundry kinds of machinery, automobile, mining spare part, and gearwheel of the engine, the driving gear of supercharger and the connceting rod. Components for power train applications, arctic composed fastener components, shafts, gears, drill collars for the oil exploration, etc. are additionally manufactured utilizing this steel alloy.

Finite element analysis (FEA) is a computerized method for predicting how a product reacts to authentic-world forces, vibration, heat, fluid flow, and other physical effects. Finite element analysis shows whether a product will break, wear out, or work the way it was designed. It is called analysis, but in the product development process, it is utilized to soothsay what is going to transpire when the product is utilized.

FEA works by breaking down an authentic object into a sizably voluminous number (thousands to hundreds of thousands) of finite elements, such as minute cubes. Mathematical equations help predict the behavior of each element. A computer then integrates up all the individual demeanors to prognosticate the comportment of the genuine object.

In this paper steel specimens of different cross sections are modeled in Autodesk Inventor and finite element analysis is done on the models by defining loads and boundary conditions for varying specimen geometry. In this way the workability of steel is studied.

2. MODELING & SIMULATION

The modeling is done using the Autodesk Inventor sketch window. Two types of models ***cylindrical and tapered*** are made for the analysis purpose.

The Fig. 2.1 and 2.2 shows the geometrical specifications for the two types of models.

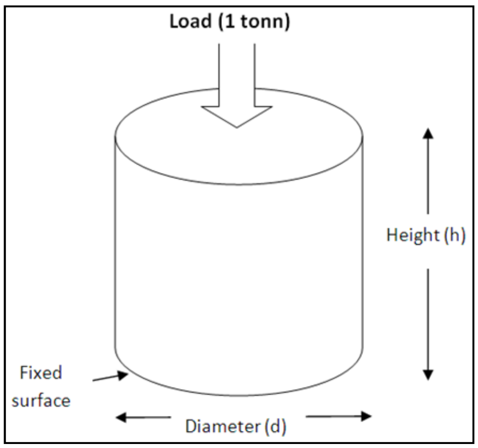


Fig. 2.1: Specifications for cylindrical model

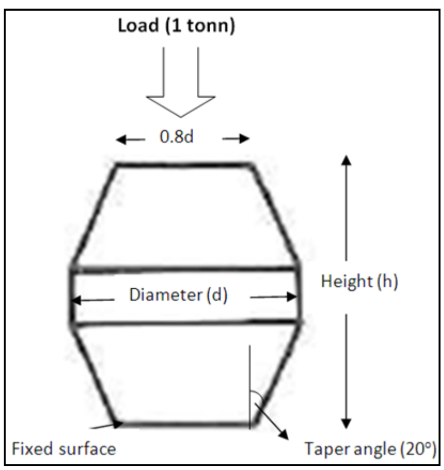


Fig. 2.2: Specifications for tapered model

For the modeling and analysis purpose the diameter of the specimen is fixed 15mm. The height of the specimen varies as per the aspect ratio. Aspect ratio is taken as ***0.75, 1, and 1.5*** for studing the deformation. The steady load applied on the upper surface of the specimen is equal to I Tonn.

Displacement of the specimen in the axial direction (direction of load application) and the von misses stress are taken as the main parameters for deformation finite element analysis. The change in the properties with respect to the aspect ratio and shape of the specimen are the major concern in this paper.

3. FINITE ELEMENT ANALYSIS

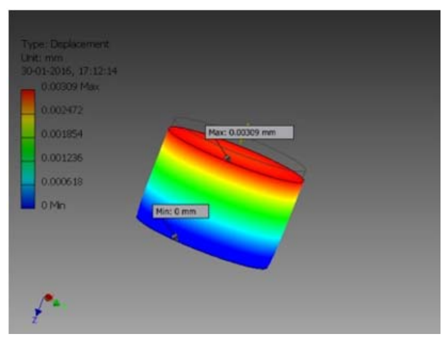
The finite element analysis is carried utilizing the Autodesk Inventor 2014 software. This simulation includes the following steps:

- The specimen of required geometry (cylindrical or tapered) is modeled in the sketch window.
- For the analysis select Environment menu →stress analysis.
- Model is transferred to the FEM analysis environment.
- The material property of the model is designated as steel (42CrMo4).
- Next, the lower surface of the specimen is constraint from any type of kineticism.
- Steady load of 1 tonn is applied on the upper surface just antithesis to the constrained surface.
- The analysis setup is consummated.
- The FEM simulation is executed for the defined boundary conditions and loads by clicking on the RUN button
- The required stress, strain and displacement results are interpreted.

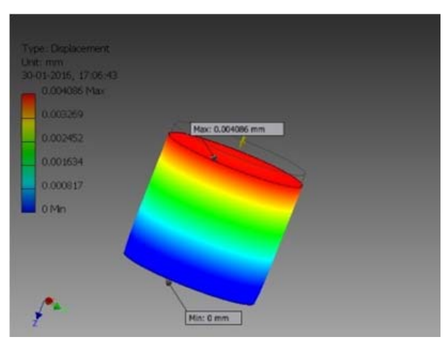
The steps are reiterated for different aspect ratio 0.75, 1 & 1.5.

4. RESULT AND DISCUSSIONS

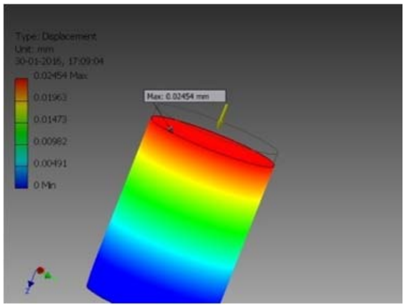
4.1 CYLINDRICAL SPECIMEN



(a)

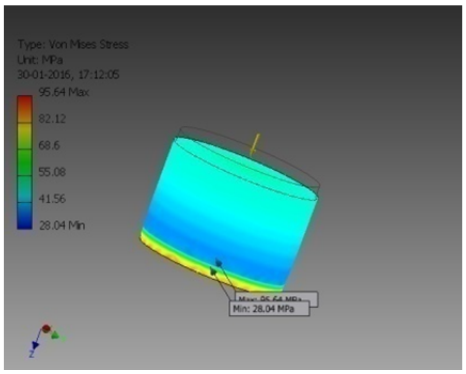


(b)

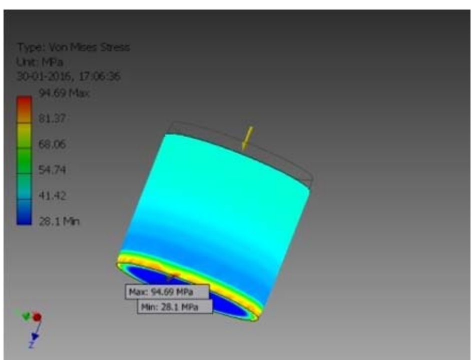


(c)

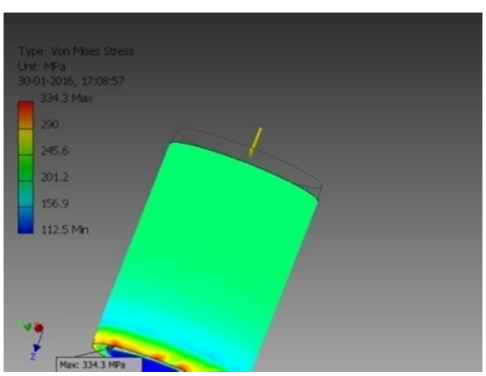
Fig. 4.1: Displacement in the cylindrical models of aspect ratio (a) 0.75, (b) 1 & (c) 1.5 resp. under application of 1 Tonn load



(a)



(b)



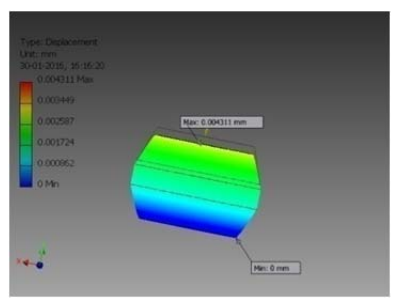
(c)

Fig. 4.2: Von misses stress variation in the cylindrical models of aspect ratio (a) 0.75, (b) 1 & (c) 1.5 resp. under application of 1 Tonn load

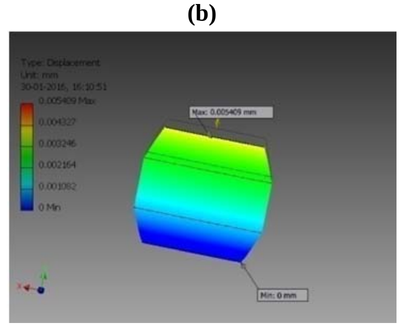
Table 4.1: Simulation results for cylindrical specimen of different aspect ratio.

Aspect ratio	0.75		1		1.5	
Name (Properties)	Minimum	Maximum	Minimum	Maximum	Minimum	Maximum
Volume	1988.04 mm^3		2650.72 mm^3		3976.08 mm^3	
Mass	0.0153675 kg		0.0204901 kg		0.0307351 kg	
Von Mises Stress	28.0413 MPa	95.644 MPa	28.1042 MPa	94.69 MPa	28.1298 MPa	83.5863 MPa
1st Principal Stress	-36.7998 MPa	2.41342 MPa	-36.1072 MPa	1.05756 MPa	-31.6949 MPa	0.864225 MPa
3rd Principal Stress	-139.464 MPa	-49.0721 MPa	-137.802 MPa	-49.1758 MPa	-121.326 MPa	-49.2089 MPa
Displacement	0 mm	0.00308997 mm	0 mm	0.00408607 mm	0 mm	0.00613582 mm
Safety Factor	2.61386 ul	8.91541 ul	2.64019 ul	8.89546 ul	2.99092 ul	8.88737 ul
X Displacement	-0.000644825 mm	0.000643864 mm	-0.000627383 mm	0.000626985 mm	-0.000628839 mm	0.000625726 mm
Y Displacement	-0.000644297 mm	0.000644504 mm	-0.000627185 mm	0.000627194 mm	-0.000627363 mm	0.000627335 mm
Z Displacement	0 mm	0.00302194 mm	0 mm	0.00403766 mm	0 mm	0.006104 mm
Equivalent Strain	0.000145192 ul	0.000455984 ul	0.000145507 ul	0.000451107 ul	0.000145621 ul	0.00039767 ul

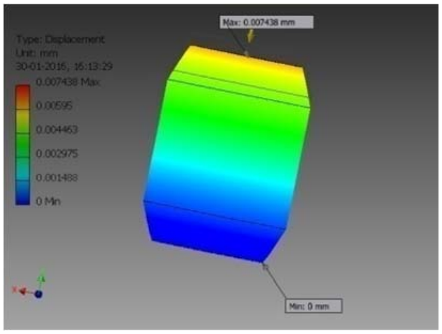
4.2 TAPERED SPECIMEN



(b)

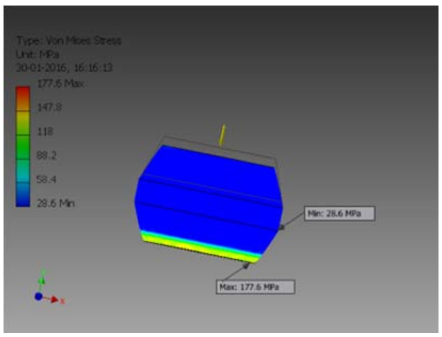


(b)

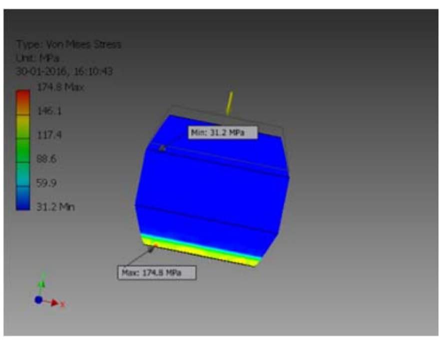


(c)

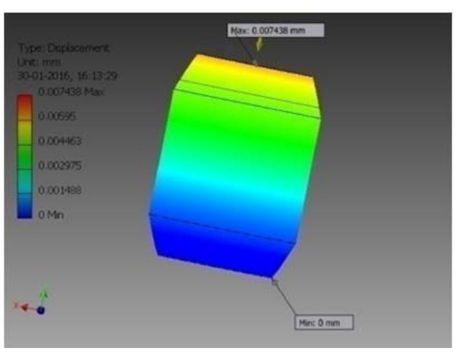
Fig. 4.3: Displacement in the tapered models of aspect ratio (a) 0.75, (b) 1 & (c) 1.5 resp. under application of 1 Tonn load



(a)



(b)



(c)

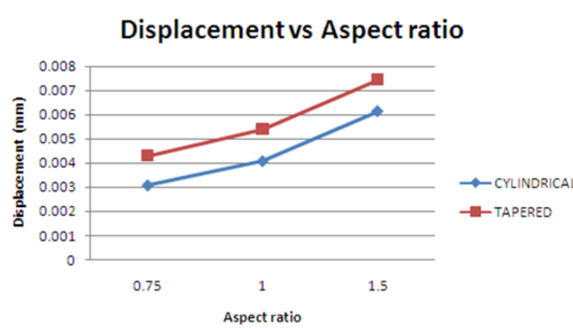
Fig. 4.2: Von misses stress variation in the cylindrical models of aspect ratio (a) 0.75, (b) 1 & (c) 1.5 resp. under application of 1 Tonn load

Table 4.2: Simulation results for tapered specimen of different aspect ratio.

Aspect ratio	**0.75**		**1**		**1.5**	
Name (Properties)	**Minimum**	**Maximum**	**Minimum**	**Maximum**	**Minimum**	**Maximum**
Volume	1726.42 mm^3		2402.88 mm^3		3755.8 mm^3	
Mass	0.0133452 kg		0.0185743 kg		0.0290323 kg	
Von Mises Stress	28.6176 MPa	177.556 MPa	31.1723 MPa	174.821 MPa	33.1783 MPa	137.006 MPa
1st Principal Stress	-109.704 MPa	16.0272 MPa	-109.079 MPa	13.406 MPa	-49.1313 MPa	11.7571 MPa
3rd Principal Stress	-292.483 MPa	-17.5803 MPa	-288.818 MPa	-22.2257 MPa	-187.501 MPa	-25.9228 MPa
Displacement	0 mm	0.00431104 mm	0 mm	0.00540925 mm	0 mm	0.00743782 mm
Safety Factor	1.40801 ul	8.7359 ul	1.43003 ul	8.01995 ul	1.82474 ul	7.53504 ul
X Displacement	-0.00080327 mm	0.000801078 mm	-0.000758963 mm	0.000763142 mm	-0.000729496 mm	0.000727609 mm
Y Displacement	-0.00431101 mm	0 mm	-0.00540924 mm	0 mm	-0.00743782 mm	0 mm
Z Displacement	-0.000802187 mm	0.000803185 mm	-0.000761685 mm	0.000760508 mm	-0.000724962 mm	0.00072893 mm
Equivalent Strain	0.000121007 ul	0.000891487 ul	0.000132032 ul	0.000878985 ul	0.000141097 ul	0.000636979 ul

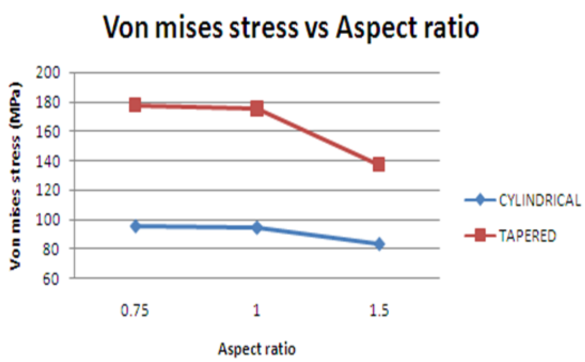
5. CONCLUSION

Dispalcement of the specimen in the z direction or the reduction of the height of the specimen during the application of 1 tonn steady load is studied for both cylindrical and tapered specimen. The graph 1 gives a clear conception about the displacement occuring due to deformation with deference to the aspect ratio.



Graph 5.1: Change in height of the specimen with respect to aspect ratio

From the above graph, we infer that the aspect ratio is directly proportional to the deformation occuring in z direcction (the direction in which 1 Tonn steady load is applied). Moreover another consequential factor on which the strain depends is the shape of the specimen. In case tapered specimen the deformation or the strain engendered is more compared to the cylindrical specimen.



Graph 5.2: Von misses stress vs aspect ratio

Another consequential parameter taken into consideration is the von misses stress. Von misses stress gives the value of the equivalent stress action during this deformation process. Graph 5.2 betokens that more the aspect ratio less will be the value of von misses stress. There is a sudden dip in the value of stress from aspect ratio 1 to 1.5. This is because more minute the specimen more energy is required for the deformation of the structure. We can interpret that the stresses acting in case of tapered specimen are higher than that in case of cylindrical specimens.

Thus, we can conclude

- Aspect ratio is customarily taken in the range of 0.75-1.5 for better results.
- The aspect ratio of the specimen has a direct impact on strain or the displacement occuring due to the application of load.
- Von misses stress is inversely proportional to the aspect ratio. More minute the specimen higher the stress applied.
- Aspect ratio is a main parameter determining the workability of a specimen.
- Shape or geometry of the specimen also plays a major role in case of workability. Tapered sections have a higher strain value compared to the cylindrical section. Stress values are higher in case of tapered sections.
- Other factors like friction, strain rate and temperature additionally affect the workability of a material

REFERENCES

[1] S. N. Shah, C. H. Lee and S. Kobayashi, "Compression of tall, circular, solid cylinders between parallel flat dies ", *Proc. Int. Conf. Prod. Engr*, Tokyo, p. 295 (1974).

[2] S. I. On and S. Kobayashi, "Workability of aluminum alloy 7075-T6 in upsetting and rolling", *Trans. ASME, J. Engr*, Ind, 98, 800 (1976).

[3] I. Fried, "Finite element analysis of incompressible material by residual energy balancing", *Int. Z Solids Structures 10*, 993 (1974).

[4] J. W. H. PRICE and J. M. ALEXANDER, "Specimen geometries predicted by computer model of high deformation forging", *Int. J. Mech. Sci*. 21, 417 (1979).

[5] C. C. Chen and S. Kobayashi, "Rigid-plastic finite-element analysis of ring compression", *Applications of Numerical Method of Forming Processes, ASME, AMD*, Vol. 28, p. 163 (1978).

[6] J. W. H. Price and J. M. Alexander, "Specimen geometries predicted by computer model of high deformation forging", Int. *J. Mech. Sci*. 21, 417 (1979).

[7] H. Kudo, "An upper-bound approach to plane strain forging and extrusion", I. Int. J. Mech. Sci.1, 57 (1960).

[8] S. Key and R. D. Krieg, "On application of the finite element method to metal forming process 1", *Comput. Meth. Appl. Mech. Engng*, 17, 597, (1978).