Design And Analysis of Flat Head Screw

K.Chithambaram[1], Dr.P.K.Palani[2]
[1, 2] Dept of Manufacturing Engineering
[1, 2] Government College of Technology, Coimbatore

Abstract- *Screw and bolted connections which are used extensively in structure connections have the characteristics of easily to be installed and disassembled. A finite element model is established to investigate the strength and characteristic of screw connections.The main abstract of this paper is to make a design of a flat head screw model using SolidWorks software by taking basic dimensions. Then the created model is transferred to the ANSYS software as a part file and various static analysis is made. As the analysis over the flat head screw is not previously done by the experts here the a simple analysis is made and the results has been concluded.*

Keywords- Finite Element Analysis, SOLIDWORKS, design parameters, meshing, total deformation, safety factor, ANSYS, equivalent stress.

I. INTRODUCTION

A screw is a combination of simple machines—it is in essence an inclined plane wrapped around a central shaft, but the inclined plane (thread) also comes to a sharp edge around the outside, which acts a wedge as it pushes into the fastened material, and the shaft and helix also form a wedge in the form of the point. A screw will usually have a head on one end that contains a specially formed shape that allows it to be turned or driven with a tool. Common tools for driving screws include screwdrivers and wrenches. The head is usually larger than the body of the screw, which keeps the screw from being driven deeper than the length of the screw and to provide a bearing surface. The cylindrical portion of the screw from the underside of the head to the tip is known as the shank; it may be fully threaded or partially threaded. The distance between each thread is called the "pitch".The majority of screws are tightened by clockwise rotation, which is termed a right-hand thread; a common mnemonic device for remembering this when working with screws or bolts is "righty-tighty, lefty-loosey". If the fingers of the right hand are curled around a right-hand thread, it will move in the direction of the thumb when turned in the same direction as the fingers are curled. Screws with left-hand threads are used in exceptional cases, where loads would tend to loosen a right handed fastener, or when non-interchangeability with right-hand fasteners is required. For example, when the screw will be subject to counterclockwise torque (which would work to undo a right-hand thread), a left-hand-threaded screw would be an appropriate choice. The left side pedal of a bicycle has a left-hand thread.

II. SOLIDWORKS

SolidWorks is a solid modeling computer-aided design (CAD) and computer-aided engineering (CAE) computer program that runs on Microsoft Windows. SolidWorks is published by Dassault Systems. SolidWorks is a solid modeler, and utilizes a parametric feature-based approach which was initially developed by PTC (Creo/Pro-Engineer) to create models and assemblies. The software is written on Parasolid-kernel.

File Format

SolidWorks files use the Microsoft Structured Storage file format. This means that there are various files embedded within each SLDDRW (drawing files), SLDPRT (part files), SLDASM (assembly files) file, including preview bitmaps and metadata sub-files. Various third-party tools (see COM Structured Storage) can be used to extract these sub-files, although the sub files in many cases use proprietary binary file formats.

Design Parameters

Circle diameter = 3 mm
Extruded length = 19 mm
Cone length = 7 mm ; Angle = 12˚
Head length = 3 mm ; Angle = 30˚
Fillet length = 68 mm
Rectangle = 1*4.5 mm2
Extruded cut = 2 mm ; Angle = 10˚
Plane = 17 mm
Pitch = 2 mm
Revolution = 10
Helix and spiral and variable pitch

Regional Parameters

S. No	Pitch (mm)	Revolution	Height (mm)	Diameter (mm)
1.	2	0	0	3
2.	2	6	12	3
3.	2	7	14	2.6
4.	2	8	16	2.4
5.	2	9	18	2
6.	2	10	20	1

Design of Screws

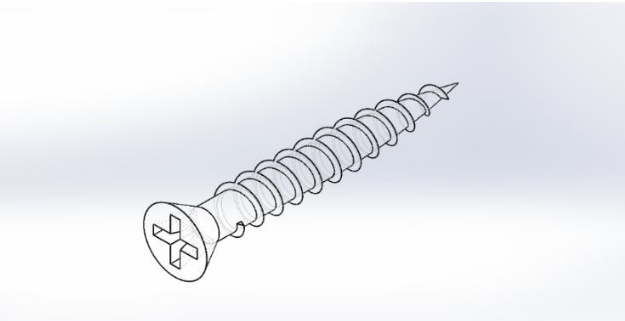

Fig.1 Flat head screw

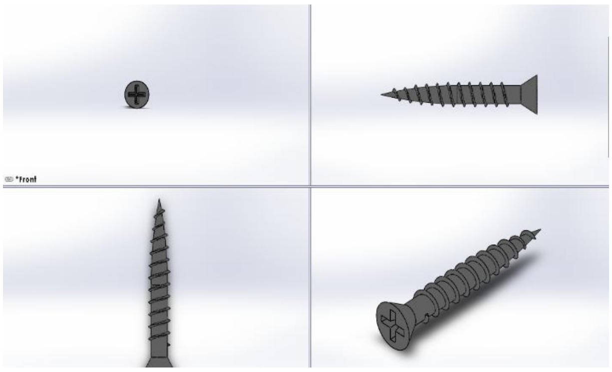

Fig.2 Various views of flat head screw

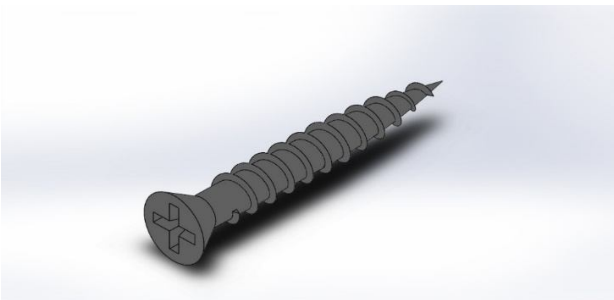

Fig.3 Isometric view

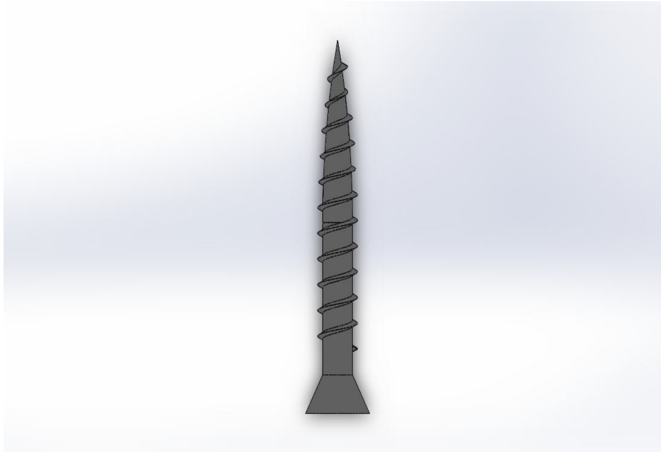

Fig.4 Screw placing over the base

III. FINITE ELEMENT ANALYSIS

The Finite Element Analysis (FEA) is a numerical method for solving problems of engineering and mathematical physics. Useful for problems with complicated geometries, loadings and material properties where analytical solutions cannot be obtained.

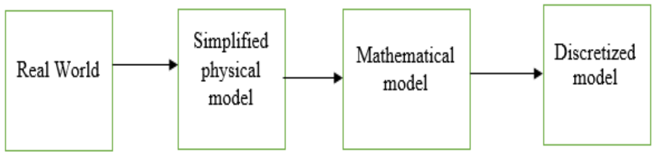

Fig.5 Process flow

A. STEPS INVOLVED:

Preprocessing

Define the geometric domain of the problem.
Define the element types
Define the material properties of the elements.
Define the geometric properties of the elements (length, area).
Define the element connectivities (mesh the model).
Define the physical constraints (boundary conditions).
Define the loadings.

Solution

Computes the unknown values of the primary field variable(s). Computed values are then used by back substitution to compute additional, Derived variables, such as reaction forces, element stresses, and heat flow.

Post Processing

Post processor software contains sophisticated routines used for sorting, printing and plotting.
Selected results from a finite element solution.

IV. RESULTS AND DISCUSSION

A. ANSYS

Ansys software is used to design products and semiconductors, as well as to create simulations that test a product's durability, temperature distribution, fluid movements, and electromagnetic properties.

B. MESHING

Mesh generation is the practice of creating a mesh, a subdivision of a continuous geometric space into discrete geometric and topological cells. Often these cells form a simplicial complex. Usually the cells partition the geometric input domain. Mesh cells are used as discrete local approximations of the larger domain.

C. STATIC STRUCTURE

A static structural analysis determines the displacements, stresses, strains, and forces in structures or components caused by loads that do not induce significant inertia and damping effects. Steady loading and response conditions are assumed; that is, the loads and the structure's response are assumed to vary slowly with respect to time.

D. TOTAL DEFORMATION

Directional deformation can be put as the displacement of the system in a particular axis or user defined direction. Total deformation is the vector sum all directional displacements of the systems.

E. EQUIVALENT STRESS

Equivalent stress (also called von Mises stress) is often used in design work because it allows any arbitrary three-dimensional stress state to be represented as a single positive stress value. Equivalent stress is part of the maximum equivalent stress failure theory used to predict yielding in a ductile material.

F. FACTOR OF SAFETY

Factor of safety (FoS) is ability of a system's structural capacity to be viable beyond its expected or actual loads. For example, a safety factor of 2 does not mean that a structure can carry twice as much load as it was designed for. The safety factor depends on the materials and use of an item.

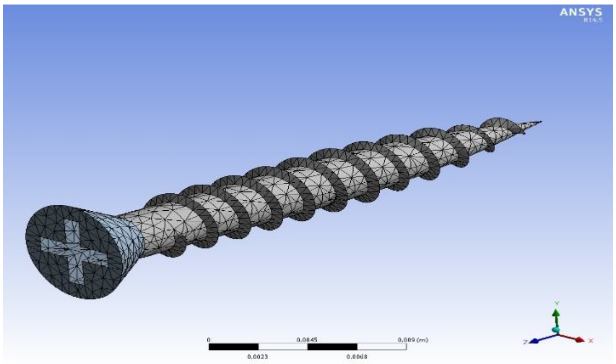

Fig.6 Meshing of the screw

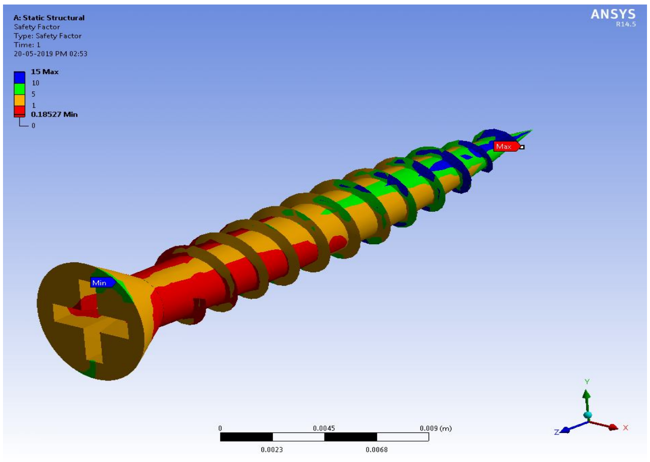

Fig. 7 Static structure

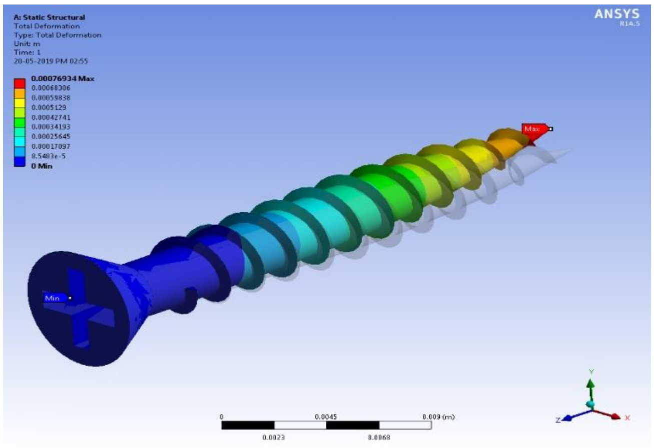

Fig. 8 Total deformation

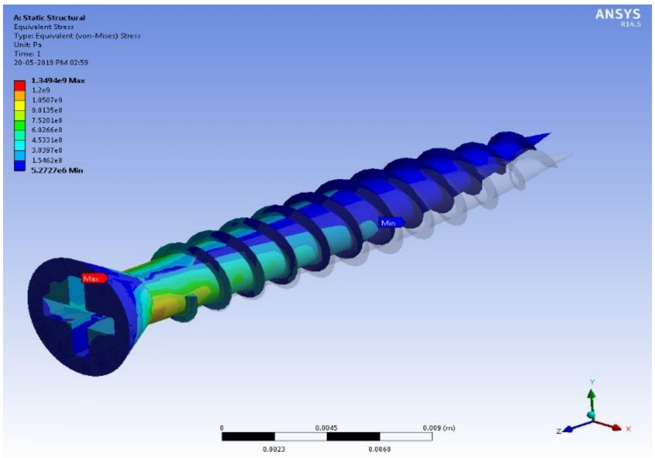

Fig. 9 Equivalent stress

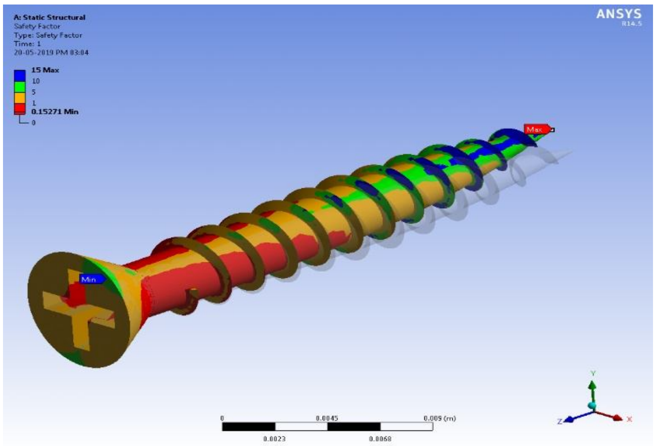

Fig. 10 Safety factor

Table 1 Value of equivalent stress

Stress	Maximum	Minimum
Equivalent stress	5.2727e6	1.3494e9

Table 2 Deformation range of the screw

Deformation	Maximum	Minimum
Total deformation	0.00076934	0

Table 3 Safety factor value

Factor	Maximum	Minimum
Safety factor	15	0.15271

CALCULATIONS

- Mean diameter, d = (do + dc) /2
- dc – minor dia; do – major dia = dc + pitch
- dc = 3 mm ; do = 3+2 = 5 mm
- d = (5+3)/2 = 4 mm

PARAMETER USED

Young's modulus of steel (E) = 2.1*10 5 N/mm^2
Poisson's ratio of steel (NU) = 0.28
Pitch=2 mm

PERMISSIBLE STRESSES FOR SCREW MATERIAL:

Permissible compressive stress FS = 2 = (360/2) = 180
Permissible shear stress calculation *y* = yield stress in shear = (0.5 to 0.6) ×*y*

ꞇ = ꞇy/ FS
= 360/4
= 90 N/mm^2

Therefore the calculated value is 90 N/mm^2
The analysis value is 93.4 N/mm^2

V. CONCLUSION

Analyzing the flat head screw using ANSYS software it was observed that in the static analysis, the equivalent stress of the power screw is within the permissible limit.

It was found that the stress analysis by conventional method is much difficult, takes a longer time and error chances are high. Ansys also allows us to find the stress throughout the section. It also helps to differentiate the stress formation in section by colour coding. The theoretical shear stresses of power screw is found manually and then analyzed in ANSYS software. The readings are shown in the tabular column. It is found that comparing with manual results, results are approximate or closer to it.

REFERENCES

[1] Xiaogang Li, XiaotianZhang ., The strength analysis of steel sunk screw connections in the rocket.
[2] Srividya T., Kaviya B., Effect on mesh reinforcement on the permeability and strength of previous concrete, International Journal of Applied Engineering Research.
[3] YanyaoJiang , Ming Zhang, Chu-Hwa Lee, " A study of early stage self-loosening of bolted joints" Journal of Mechanical Engineering.
[4] Jorge Angeles, The virtual screw: Concept, design and applications.
[5] Joseph I. Orisaleye, Parametric analysis and design of straight screw extruder for solids compaction.

[6] N.LeninRakesh, V.Joseananth vino., Analysis of stress square thread power screws using Ansys. Volume 116 No. 14 2017, 455-460.
[7] A. Thirugnanam, Amit Kumar and Lenin Rakesh., Analysis of Power Screw Using ‘Ansys’ Middle-East Journal of Scientific Research 20 (6): 742-743, 2014.
[8] Thomas Jeandin, Christian Mascle ., A new model to select fasteners in design for disassembly .
[9] Abbas Mohajeranin, DusanBosnjak, Damon Bromwich Analysis and design methods of screw piles: A review
[10] Srinivasan V., Analysis of static and dynamic load on hydrostatic bearing with variable viscosity and pressure, Indian Journal of Science and Technology.