International Journal of Applied Research 2016; 2(2): 01-03

ISSN Print: 2394-7500
ISSN Online: 2394-5869
Impact Factor: 5.2
IJAR 2016; 2(2): 01-03
www.allresearchjournal.com
Received: 01-12-2015
Accepted: 03-01-2016

Silviya Deckova
Assistant Profesor,
Department MEMETE,
Technical University of Sofia,
Engineering Pedagogical
Faculty - Sliven, Sliven,
Bulgaria, European Union

Tihomir Dobrev
Student in Master's degree,
Department MEMETE,
Technical University of Sofia,
Engineering Pedagogical
Faculty - Sliven, Sliven,
Bulgaria, European Union

Ventsislav Dimitrov
Assistant Profesor,
Department MEMETE,
Technical University of Sofia,
Engineering Pedagogical
Faculty - Sliven, Sliven,
Bulgaria, European Union

Correspondence
Ventsislav Dimitrov
Assistant Profesor,
Department MEMETE,
Technical University of Sofia,
Engineering Pedagogical
Faculty - Sliven, Sliven,
Bulgaria, European Union

Design of Fitness Multi Station FMS-TD 01 Part I Construction and deformation study using on finite element method (FEM)

Silviya Deckova, Tihomir Dobrev, Ventsislav Dimitrov

Abstract
This papers proposed construction of Fitness Multi Station FMS-TD 01. The elements are designed in an environment of CAD systems Top Solid v.6.15, after that the model is converted into Solid Works 2015. Made analysis of stress and strain condition by FEM. For a numerical solution is used a software application Simulation Xpress of the SolidWorks. Theoretically defined by stress von Mises σ[MPa] and deformations (displacement) f[mm] of the general elements of the system, at a maximum load of 1000 [N].

Keywords: Finite Element Method (FEM), CAD System, Multi Gym Equipment, Fitness Multi Station, Top Solid, Solid Works

Introduction

Fitness systems, according to its structure is divided into three series - heavy, light and Street. Each of these groups is divided into different types. The types are defined

according based on load of individual muscle groups. Appliances legs, back, chest, hands, arms and midriff. Some of them can be combined in one device, with the aim of greater practicality. The combined devices are suitable for home.

System in the heavy series is most commonly used in gyms, which are used by large numbers of people. Designed for heavy duty, robust construction, and use the most - often load a muscle group.

In Lightweight series fitness system are most often used at home, for loading of several muscle groups. To this group belongs, and the system proposed in this papers.

hence The purpose of this work is to design and simulate Fitness Multi Station FMS-TD 01. An analysis of stress and strain state of sports equipment in the LEC.

Street series, is used in systems for training without weights, therefore, their purpose is not to build muscle mass, but exercise and healthy living.

The objective of this papers is to design and simulate Fitness Multi Station FMS TD-01. For analysis of the stress and strain condition of this sports system using on FEM.

2. Design on Fitness Multi Station FMS-TD 01

In this paper is proposed construction of Fitness Multi Station. In the most common type of scheme Fitness Station is shown in Figure 1.

2.1 Creating a spatial model of Fitness Multi Station FMS-TD 01

The device is built in an environment of CAD systems TopSolid v.6.15. Fig. 2 shows the three-dimensional models of components and assemblies. The elements used in the manufacture of materials are selected according to their functional purpose:

- plates – St3 – sheet,
- for rollers – 40X – high strength and hardness after heat treatment,
- for caps and plates – St45 economical method for preparing a preform and easy workability of the surfaces,
- profiles – 08kp.

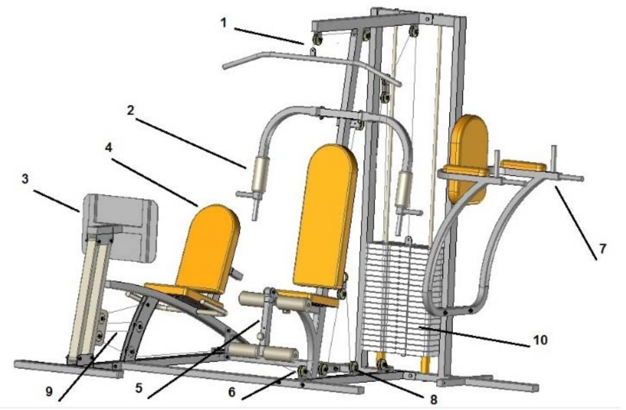


Fig 1: Fitness Multi Station FMS-TD 01 – general view
1. Top pulley handle, 2. Pec Deck, 3. Leg Press, 4. Seat, 5. Seated Leg Curls, 6. Bottom pulley handle, 7. Fitness parallel bars, 8. Rollers, 9. Steel ropes, 10. Weight 100 kg (10x10kg)

The purpose is realized on three separate stages.

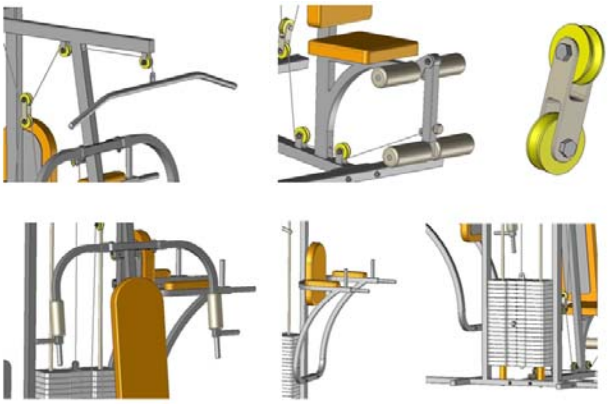
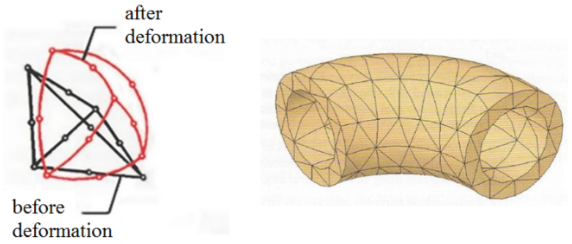


Fig 2: Three-dimensional models of elements from Fitness Multi Station FMS-TD 01

2.2 Дефиниране на модел от крайни елементи

Here are defined boundary and loading conditions and numerical experiments are carried out. After building the model is converted into SolidWorks where it is subjected to analysis by using FEM by FEM module Simulation Xpress [3,5]. In the model of system used elements of the second row, forming a parabolic field moving from the second row and subsequently, a derivative of a vector field move from the first row. Each tetrahedral element of the second row has ten nodes (four top-six internal) and each of them has three degrees of freedom. The edges and surfaces of elements of the second row accept curvilinear forms after deformation - Fig. 3,a. In the analysis using elements from the second row, using less network density in comparison with first-order items. They are modelled in accordance with curved forming the model ‐ fig. 3 b [2,4].



a) b)

Fig 3: Tetrahedral elements from the second row
a) deformation, b) meshing

Degrees of freedom of a node in a network with a finite number of elements determine the possibility of the assembly to perform translations. The number of degrees of freedom that a given node has depends on the type of element to which it belongs node.

Degrees of freedom and the forces are linked by a system of basic equations. The objective is to determine the solution of these equations based on the boundary conditions existing in the engineering object that is analyzed.

In the static analysis of the actual form of the basic equation is determined by the type [2]:

$$[\mathrm{K}]\{u\}=\{F\} \quad (1)$$

where:

- [K] stiffness matrix,
- {u} displacement vector,
- {F} force vector.

2.2.1 Setting the boundary conditions

For the purposes of the study are suitable conditions of type contact without penetration "no penetration" and provided "bonded". Contact "no penetration" is used in tubular cylindrical connections, as does not allow penetration of an interconnected series of two associated bodies in their area. The second contact "bonded" is deployments at contact surfaces that are connected, work and deformed together. The software creates a compatible network (mesh) of final elements in all contact areas. The nodes of the finite element belonging to the different parts at the contact spots coincide with each other. The selected contact conditions allow reporting of friction between the parts in their deformation. In the static analysis program using coefficient of friction established in the properties of the analysis. When defining a contact without penetration (no penetration), when a static non-linear analysis explores and is assigned a coefficient of friction that is equal to or smaller than 0.5 [2].

2.2.2 Setting the load

Realized by concentrated forces "Force" – distributed by specified indicated surfaces.

Performed static analysis, enabling visualization and systematization of tabulated results to the strength and deformation analysis of the model. In the tensile strength test are traced tensor components of the voltage and the equivalent stresses von Mises, which are calculated by the following formula (1).

$$\sigma_{eq}=\sqrt{0,5\left[(\sigma_1-\sigma_2)^2+(\sigma_2-\sigma_3)^2+(\sigma_3-\sigma_1)^2\right]} \quad (2)$$

where:

- σ_1, σ_2, σ_3 - main tensions in the three axes.

In deformation analysis are displayed on the node moves all three coordinate axes and the deformed shape of the system.

For the determination of stresses and displacements at any point of the structure using the basic laws of mechanics of the solid deformable body to draw up a system of algebraic equations of the form [2]:

$$c_{11}x_1+c_{12}x_2+\ldots+c_{1n}x_n=P_1$$
$$c_{21}x_1+c_{22}x_2+\ldots+c_{2n}x_n=P_2$$
$$\ldots$$
$$c_{n1}x_1+c_{n2}x_2+\ldots+c_{nn}x_n=P_n \quad (3)$$

where:

- $x_1, x_2, \ldots, x_n$ - are unknown (displacements and rotations)

- c_{ij} **-** coefficients having sense of reactions from single displacements and rotations

- P_i **-** external forces and moments in the corresponding direction.

2.3 Systematization and analysis of the results

Made static analysis, enabling visualization and systematization of tabulated results for the model. In strength analysis traced elements of the tensor tension and equivalent stresses von Mises. In deformation analysis are shown displacements of nodes on the three axes [1].

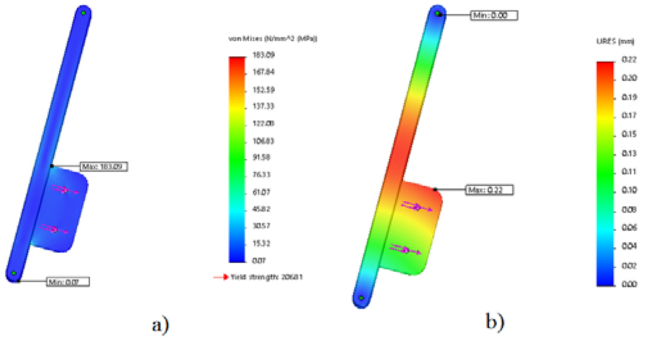


Fig 4: Stress – strain analysis of lever mechanism
a) stress, b) deformation

In Fig. 4,a and 4,b it is shown that distribution of von Mises stress and maximum displacements of lever mechanism. The high values of maximum stress is obtained in the area of connecting the two elements. They reach 183.69 Mpa and maximum displacements reached 0.22 mm. This allows to conclude that the construction meets the requirements in terms of stresses and displacements.

On fig.5,a and 5,b is shown the distribution of stresses and displacements respectively maximum of parallel bars. Most - high maximum stress are obtained in the bottom of the parallel bars. They reach 41.84 MPa and the maximum displacements reaches 0,66 mm. This allows to conclude that the construction meets the requirements in terms of stresses and displacements.

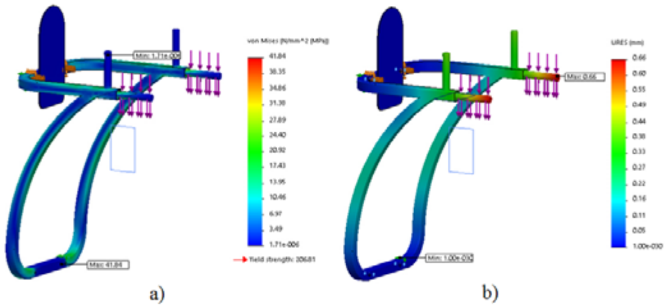


Fig 5: Stress – strain analysis of parallel bars
a) stress, b) deformation

In Figure 6,a and 6,b shows the distribution of stresses and displacements of the weights stand. Top - high values of maximum stress are obtained at the bottom of the stand. They reach 36.60 MPa and maximum displacements reach 1,15 mm. This cover requirements.

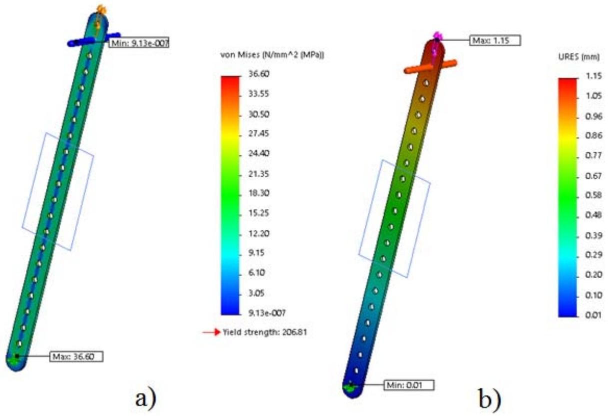


Fig 6: Stress – strain analysis of weights stand
a) stress, b) deformation

3. Conclusions

3.1 From the following results, it is clear that Fitness Multi Station FMS-TD 01 is very suitable for home and small gyms in hotels and holiday homes. It can be trained about all the muscle groups of the body. Need of purchasing separate system exercises that are many times -high price.

3.2 Stiffness of the model can be tested in different sizes and distribution loads. Analytically this is very difficult and inaccurate, and experimentally for parts with considerable weights and dimensions – practically impossible.

3.3 The analysis proves viability of constructed elements. The reported stresses and deformations are low compared to the maximum.

4. References

1. Kishore YR, Babu PM. Design and stress analysis of disc wheel using FEA, International Journal of Applied Research. 2016; 2(1):106-109.
2. Lee Huei-Huang. Mechanics of Materials Labs with SolidWorks Simulation 2013, SDC Publications, 2013.
3. Solid Works - Feature Manager, Solid Works Corporation, 2006, 2008.
4. Solid Works Simulation – Training Manual, Dassault Systemes Solid Works Corporation, Massachusetts, USA, 2009.
5. Стойчев Г, Метод на крайните елементи. Издателство Техника, София, 2000.