IMK-14 – Istraživanje i razvoj u teškoj mašinogradnji 24(2018)3, SR83-89
UDC 621 ISSN 0354-6829

Dinamička analiza mehaničke dvostubne dizalice korišćenjem kombinovanog konačno-elementnog (ANSYS) i analitičkog metoda

Rade Vasiljević[1*]
[1]Kosančićeva 1A/4, 31109 Užice (Srbija)

U ovom radu je razmatrana dinamička analiza mehaničke dvostubne dizalice korišćenjem kombinovanog konačno-elementnog i analitičkog metoda. U prvom delu rada je formiran i rešen konačno-elementni model razmatrane dizalice. Ovde je prikazana primena metoda konačnih elemenata za određivanje naponsko deformacionog stanja konstrukcije mehaničke dvostubne dizalice. Korišćen je programski paket ANSYS. U drugom delu rada su uprošćavanjem konačno-elementnog modela dizalice formirani i rešeni njeni dinamički modeli. Dobijene su dinamičke karakteristike nestacionarnih režima kretanja usled rada mehanizma za dizanje.

Ključne reči: Mehanička dvostubna dizalica, Dinamička analiza, CAD/CAE, ANSYS, Analitički metod

1. UVOD

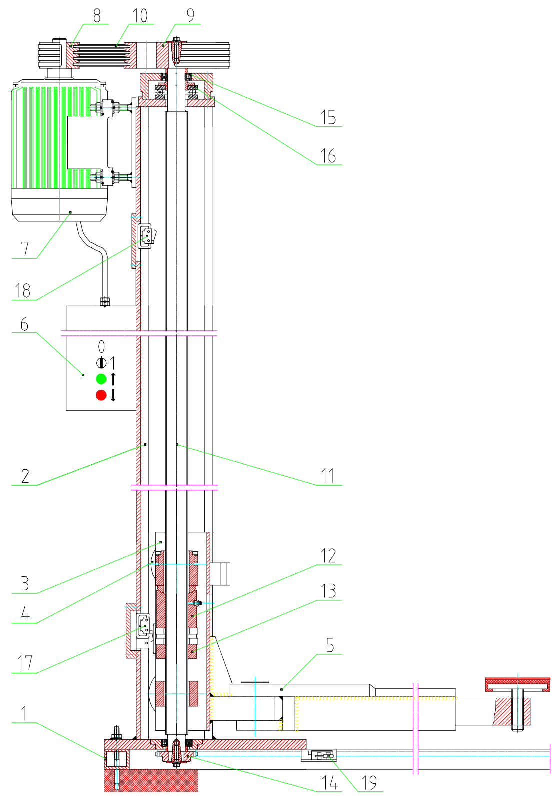
Danas dizalice treba da zadovolje spektar stavki projektovanja, gde bitno mesto zauzimaju dinamičke karakteristike dizalice. Dinamička analiza dizalica je veoma značajna, jer ona omogućava da se ustanove stvarna dešavanja u opterećenoj strukturi dizalice [1], [2]. Drugim rečima, dinamička analiza dizalica omogućava da se pri projektovanju dizalice preciznije odredi naponsko stanje i deformacije elemenata mehanizma i noseće strukture. Analizom dinamičkih karakteristika dizalice ostvaruje se približavanje stvarnom naponskom stanju elemenata dizalice, što omogućava dobijanje optimalne konstrukcije dizalice.

Pri istraživanju statičkog i dinamičkog ponašanja mehaničke dvostubne dizalice (slika 1) [3], poseban problem predstavlja složena interakcija nosećih stubova i mehanizma za dizanje (slika 2). Krutost pogonskog i pratećeg stuba jednaka je superpoziciji krutosti nosećih stubova i navojnih vretena, pri čemu su stubovi konzolno vezani za nosač dizalice, dok su navojna vretena preko ležajeva gredno smeštena u stubovima. Analitički način određivanja uticajnih koeficijenata je složen, tako da pri statičkoj i dinamičkoj analizi mehaničke dvostubne dizalice prvi korak ne treba da bude formiranje analitičkih modela.

Danas, savremeno projektovanje i konstruisanje u mašinskom inženjerstvu je nezamislivo bez primene računara [4], [5], [6]. Sa jedne strane, 3D CAD je postao standard. Sa druge strane, izrada geometrije mehaničke dvostubne dizalice ne troši mnogo vremena i isplativa je jer je ova dizalica serijski proizvod. Takođe, izrađena geometrija u 3D CAD softveru se može iskoristiti za statičku i dinamičku analizu. To znači, da prvi korak u statičkoj i dinamičkoj analizi mehaničke dvostubne dizalice treba da bude izrada 3D modela u specijalizovanom programskom paketu za 3D modeliranje, a zatim određivanje statičkih i dinamičkih parametara u specijalizovanom softveru za konačne elemente. Metod konačnih elemenata (MKE) spada u grupu savremenih numeričkih metoda [5]. Može se primeniti za statičku i dinamičku analizu složenih delova i sklopova [4], [6], [7].

Za objekat je izabrana mehanička dvostubna dizalica tipa DB2 ("Univerzal" - Banja Luka) [8].

Slika 1: Tipična mehanička dvostubna dizalica [3]

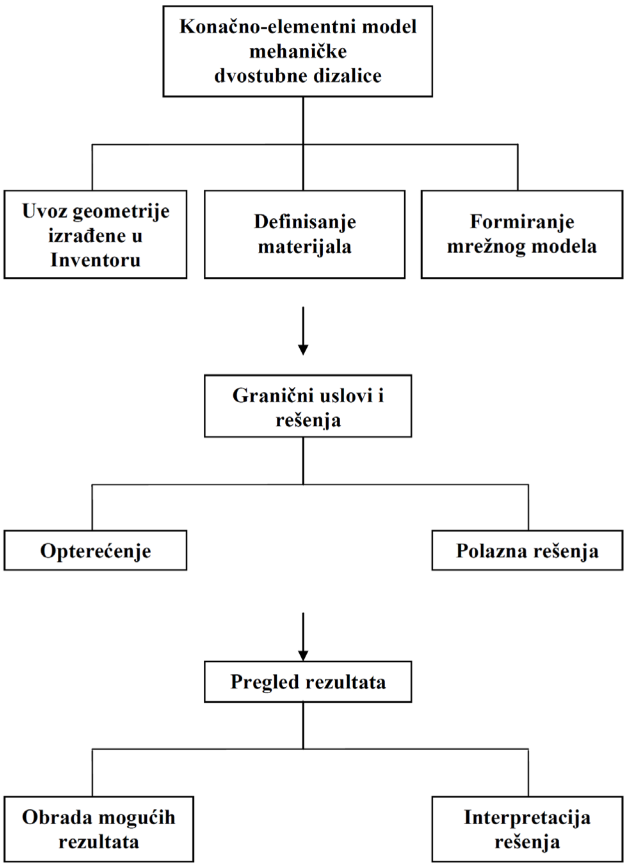


Slika 2: Pogonski stub mehaničke dvostubne dizalice

*Kontakt adresa autora: Kosančićeva 1A/4, 31109 Užice, email:r.r.vasiljevic@gmail.com

2. KONAČNO-ELEMENTNA ANALIZA - KEA (ANSYS)

Strukturni dijagram rešavanja konačno-elementne analize mehaničke dvostubne dizalice u ANSYS programu prikazan je na slici 2. Analiza prikazanog strukturnog dijagrama je data u referenci [5].



Slika 2: Strukturni dijagram KEA u ANSYS-u

2.1. Formiranje konačno-elementnog modela

Izrađen je originalni 3D solid model izvedenog rešenja mehaničke dvostubne dizalice, tipa DB2, proizvođača "Univerzal" iz Banja Luke [8], primenom programskog paketa Autodesk Inventor (CAD okruženje), slika 3. Tehnologija izrade ovog 3D modela je izložena u referenci [5].

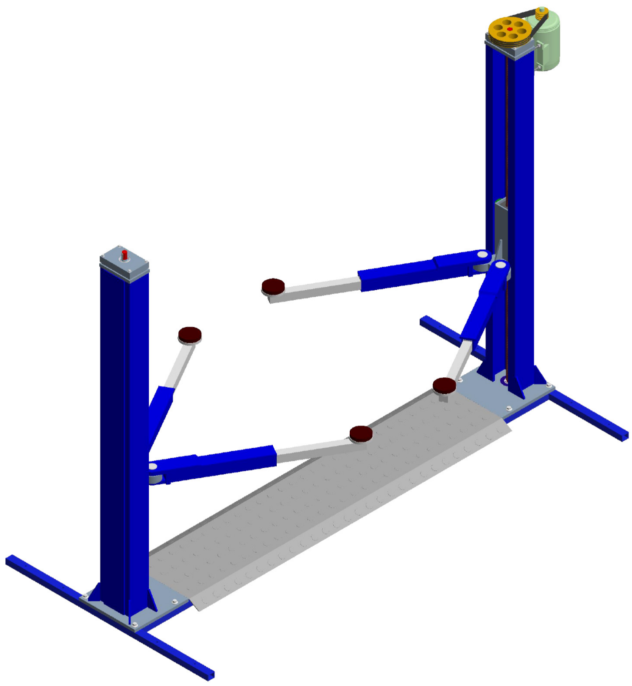
Za potrebe određivanja statičkih i dinamičkih karakteristika izrađeni 3D solid model je uvezen u programski paket ANSYS, slika 4.

Na formiranom 3D modelu, saglasno preporuci proizvođača, postolje dizalice je preko anker vijaka vezano za betonsku ploču dimenzija 3500×1600×180 [mm].

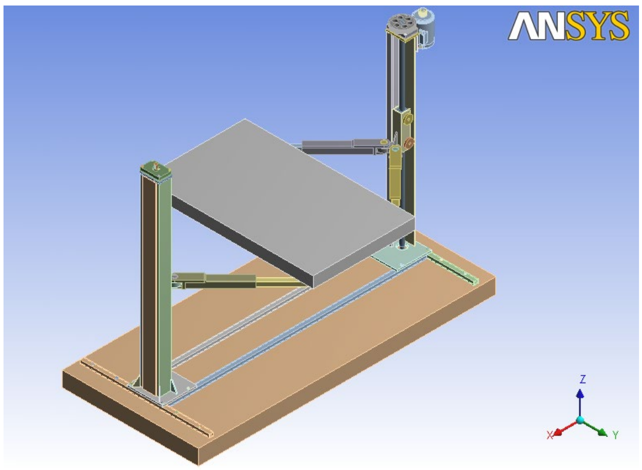
Kod ovog rešenja 3D modela mehaničke dvostubne dizalice navojna vretena su u gornjem kućištu radijalno i aksijalno fiksirana, dok su u donjem kućištu radijalno fiksirana a aksijalno slobodna.

Opterećenje dizalice je predstavljeno sa homogenom čeličnom pločom mase $m_Q = 2500$ [kg].

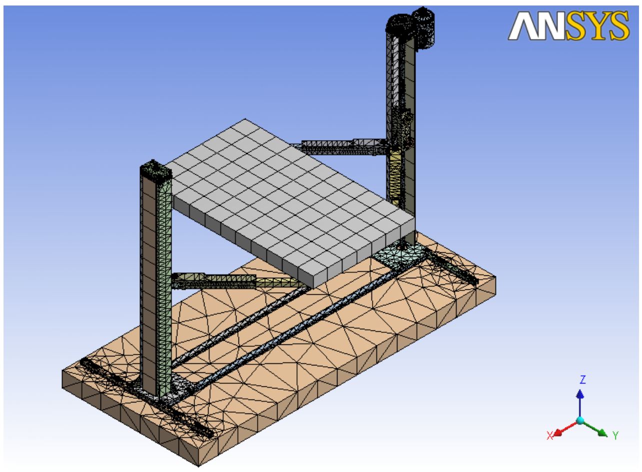
Na osnovu izrađene geometrije mehaničke dvostubne dizalice, u programu ANSYS je generisana mreža konačnih elemenata, slika 5.



Slika 3: 3D model mehaničke dvostubne dizalice - Inventor



Slika 4: 3D model mehaničke dvostubne dizalice - ANSYS



Slika 5: Generisanje mreže KE 3D modela mehaničke dvostubne dizalice - ANSYS

Na formiranom KE modelu, primenom programskog paketa ANSYS (CAE okruženje) [7], sprovedena je konačno-elementna analiza. Analizirana su tri karakteristična položaja pokretnih nosača konzola:

- I - donji krajnji položaj pokretnih nosača konzola,
- II - srednji položaj pokretnih nosača konzola, i
- III - gornji krajnji položaj pokretnih nosača konzola.

2.2. Statička strukturna analiza

Kako noseće konstrukcije, tako i navojnih vretena analiziran je karakter i maksimalne vrednosti sledećih statičkih karakteristika:

- statičkih pomeranja, i
- statičkih napona.

2.2.1. Statička pomeranja

Iz uporedne analize pomeranja za tri karakteristična slučaja dobijeno je da su maksimalna pomeranja na mestu papuča konzola.

Pomeranja papuča konzola su neznatno veća u krajnjem gornjem položaju u odnosu na srednji položaj pokretnih nosača konzola ($\delta_{pk,II}$=5.92 [mm], $\delta_{pk,III}$=6.04 [mm]).

Pomeranja vrhova stubova su najveća za srednji položaj pokretnih nosača konzola i to kod pratećeg stuba ($\delta_{vs,II}$=0.78 [mm]).

Navojna vretena, saglasno promeni krutosti, najveći ugib imaju u srednjem položaju pokretnih nosača konzola ($\delta_{nv,II}$ =0.295 [mm]).

Poređenjem vrednosti pomeranja za sva tri položaja može se zaključiti da je dizalica najosetljivija na statičke uticaje kada se pokretni nosači konzola nalaze u srednjem položaju (II). To znači, da je sa aspekta statičkih pomeranja, srednji položaj pokretnih nosača konzola kritični položaj.

Na slikama 6 i 7 su prikazana pomeranja dizalice kada je opterećena maksimalnom masom tereta u gornjem krajnjem položaju.

2.2.2. Statički naponi

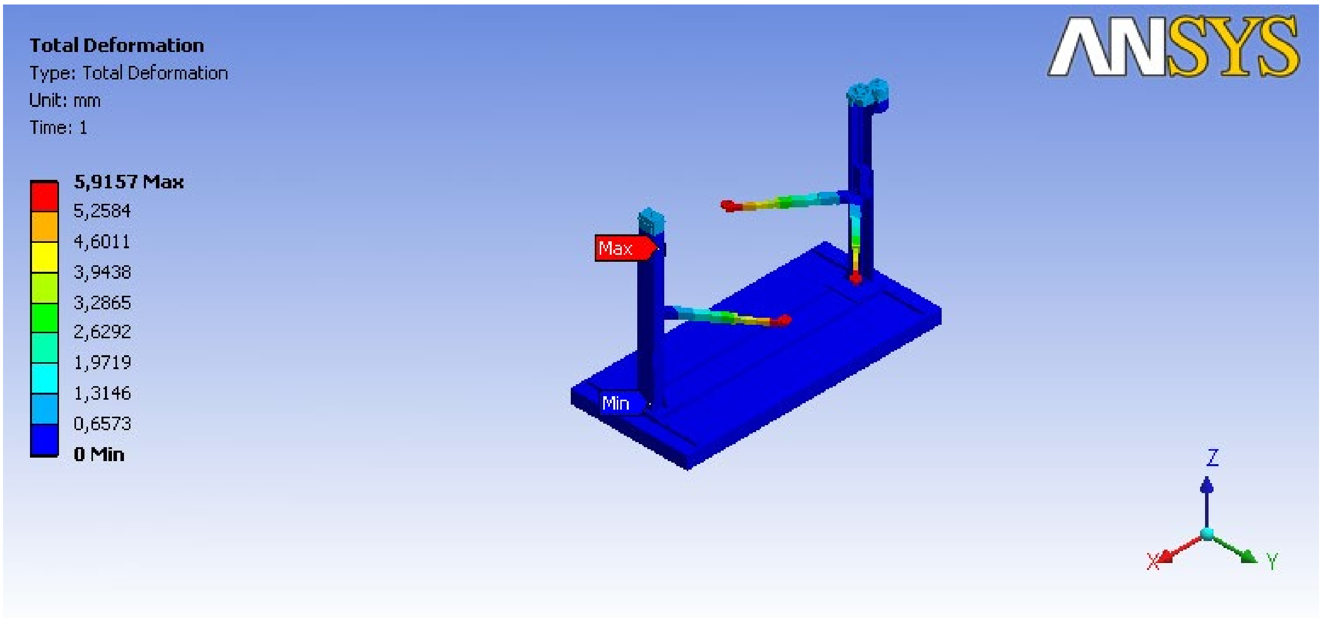
Analogno analizi pomeranja, iz uporedne analize normalnih napona za tri karakteristična slučaja vidi se da se maksimalni normalni naponi nalaze u zoni pokretnog nosača konzola.

Normalni naponi su u zoni pokretnog nosača konzola neznatno veći u krajnjem gornjem položaju u odnosu na srednji položaj pokretnih nosača konzola ($\sigma_{pn,II}$=508.38 [MPa], $\sigma_{pn,III}$=540.83 [MPa]).

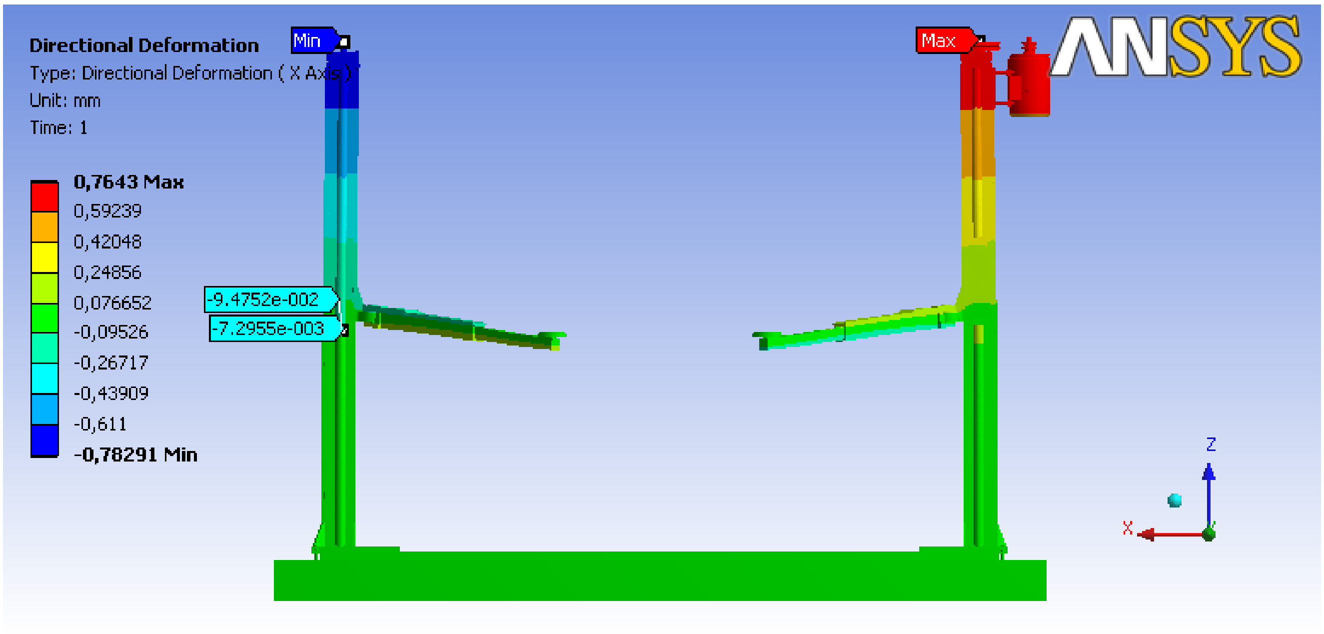
Navojna vretena, saglasno promeni krutosti i dobijenoj raspodeli napona, imaju koncentraciju većih normlnih napona u srednjem položaju pokretnih nosača konzola. Posledično, noseći stubovi imaju koncentraciju većih normalnih napona za srednji položaj pokretnih nosača konzola.

Takođe, na osnovu raspodele napona u zoni interakcije vodećih točkova pokretnih nosača konzola i nosećih stubova, opaža se postojanje lokalnog savijanja, koje je veće za srednji položaj pokretnih nosača konzola.

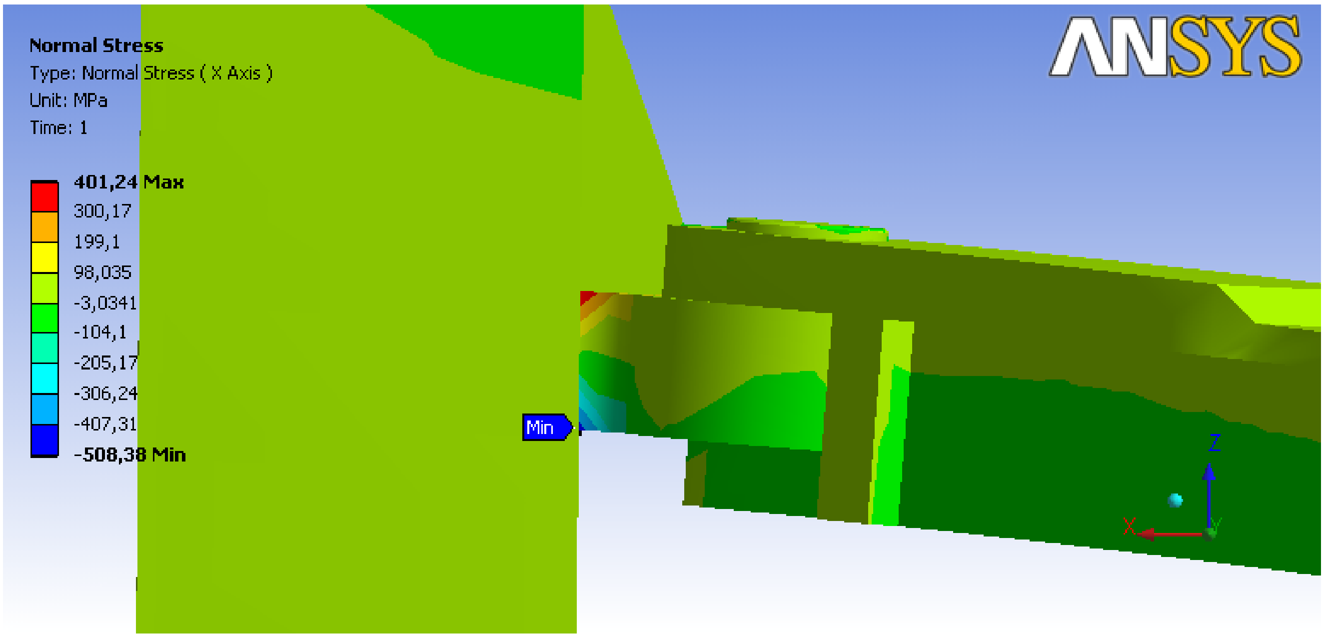
Na slici 8 je prikazan detalj raspodele napona u zoni maksimalnih napona za konstrukciju dizalice, kada je opterećena maksimalnom masom tereta u srednjem položaju.



Slika 6: Totalna pomeranja mehaničke dvostubne dizalice



Slika 7: Pomeranja u pavcu X mehaničke dvostubne dizalice



Slika 8: Detalj raspodele statičkih napona u zoni maksimalnih napona za konstrukciju dizalice

2.3. Modalna analiza

Na efikasan i pouzdan rad mehaničke dvostubne dizalice utiču sopstvene frkvencije. Sopstvene frekvencije su u direktnoj zavisnosti od krutosti i mase konstrukcije. Određivanje sopstvenih frekvencija je prvi korak u sprovođenju dinamičke analize mehaničke dvostubne dizalice.

Numeričko određivanje modalnih parametara sprovodi se metodom konačnih elemenata, primenom programskog paketa ANSYS [7]. Analiziraju se sledeći modalni parametri:

- frekvencije oscilovanja, i
- oblici oscilovanja.

2.3.1. Frekvencije oscilovanja

Za sva tri položaja pokretnih nosača konzola, određeno je prvih šest frekvencija oscilovanja 3D modela, što je podrazumevani broj u ANSYS-u.

Poređenjem vrednosti frekvencija za sva tri slučaja može se zaključiti da je dizalica dinamički najosetljivija sa aspekta prve najniže (osnovne) frekvencije u slučaju kada se pokretni nosači konzola nalaze u gornjem krajnjem položaju. Ali kako se osnovna frekvencija u krajnjem gornjem položaju vrlo malo razlikuje od osnovne frekvencije u srednjem položaju, sa jedne strane, i kako je krutost navojnog vretena najmanja u srednjem položaju i pomeranje vrhova stubova dizalice najveće u srednjem položaju, sa druge strane, položaj II je najmerodavniji za dalje opisivanje dinamičkog stanja konstrukcije.

Prvih šest frekvencija, kada je dizalica opterećena maksimalnom masom tereta u srednjem položaju, dobijenih primenom programskog paketa ANSYS, prikazano je u tabeli 1.

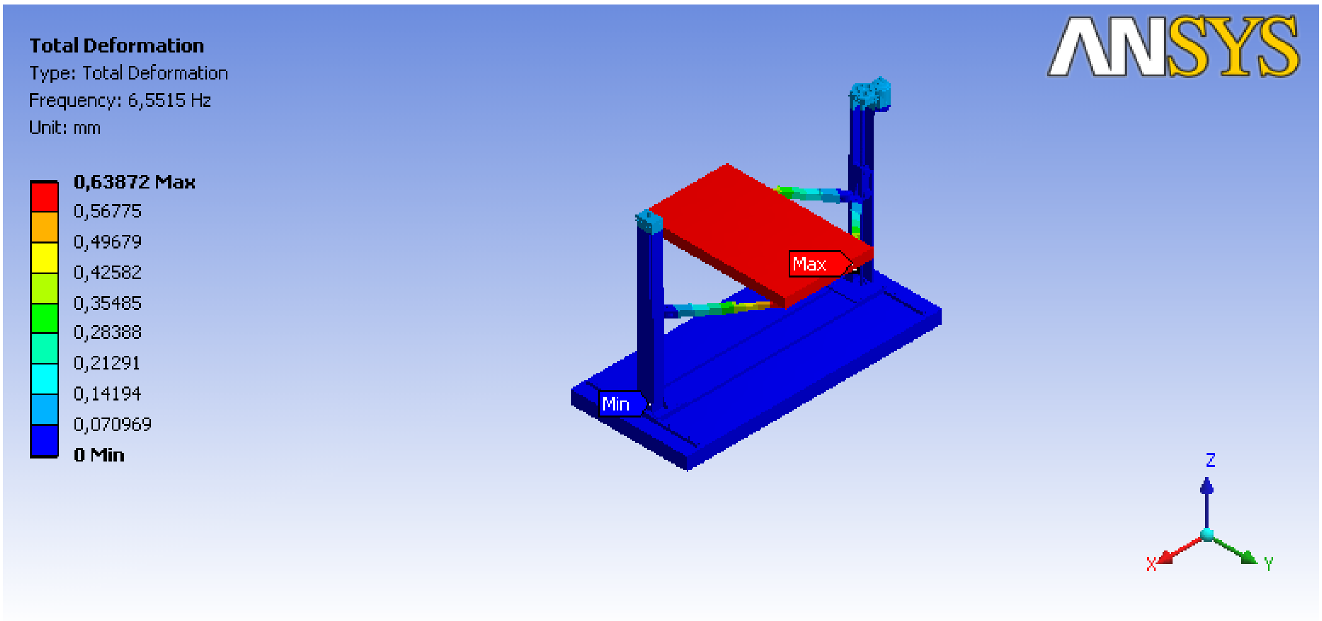
Tabela 1: Frekvencije

Mod	Frek., f [Hz]	Kr. frek., ω [s^{-1}]	Period, T [s]
1	6.55	41.14	0.153
2	8.25	51.83	0.121
3	11.12	69.85	0.09
4	14.56	91.42	0.069
5	14.79	92.90	0.068
6	23.84	149.73	0.042

2.3.2. Forms of oscillation

Modalnom analizom dobijeno je prvih 6 oblika oscilovanja mehaničke dvostubne dizalice za srednji položaj pokretnih nosača konzola, kada je dizalica opterećena maksimalnom nosivošću. Na slici 9 je prikazan prvi glavni oblik oscilovanja.

Sa slike 9 se uočava da oblik oscilovanja u pravcu dizanja i spuštanja motornog vozila (1 mod) odgovara elastičnoj liniji u vertikalnoj ravni, odnosno deformisanom obliku usled zadržavanja motornog vozila na određenoj visini dizanja. Takođe, uočava se dominantnost oscilovanja konzola u vertikalnom pravcu, u odnosu na druge elemente konstrukcije.

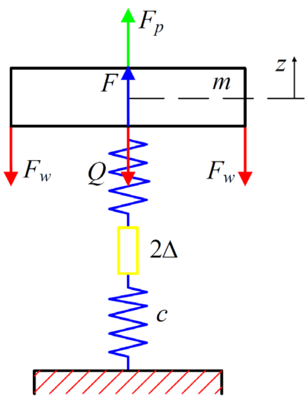


Slika 9: 1. mod oscilovanja mehaničke dvostubne dizalice, T_1=0.153 [s]

3. ANALITIČKA ANALIZA NESTACIONARNIH KRETANJA

Na osnovama konačno-elementne analize (u prethodnom podnaslovu), sprovedeno je analiziranje dinamičkih karakteristika dizalice u nestacionarnim režimima usled rada mehanizma za dizanje.

Uprošćavanjem 3D modela dizalice, formirani su ekvivalentni analitički dinamički modeli, slike 10 i 11, tako da su isti zadržali osnovnu frekvenciju oscilovanja noseće strukture u vertikalnom pravcu. Za formiranje ovih matematičkih modela kao podloga korišćeni su statički i dinamički parametri dobijeni statičkom strukturnom i modalnom analizom u ANSYS-u. Formiranju ovih modela predhodio je pregled referenci [1], [2], [9], [10], [11].



Slika 10: Dinamički model mehaničke dvostubne dizalice za režim ubrzanja pri dizanju

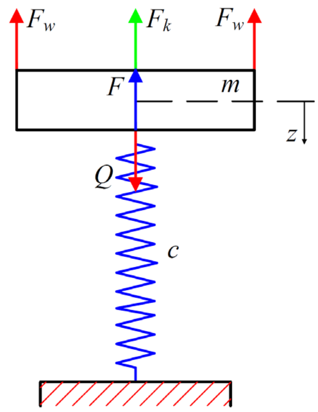


Figure 11: Dinamički model mehaničke dvostubne dizalice za režim kočenja pri spuštanju

Diferencijalne jednačine kretanja dinamičkih modela, koje opisuju dinamičko ponašanje razmatrane dizalice u vertikalnoj ravni u nestacionarnim režimima usled rada mehanizma za dizanje motornog vozila, glase:

$$m \cdot \ddot{z} + c \cdot z = F_p - m \cdot g - 2\,F_w \qquad (1)$$

$$m \cdot \ddot{z} + c \cdot z = m \cdot g - F_k - 2\,F_w \qquad (2)$$

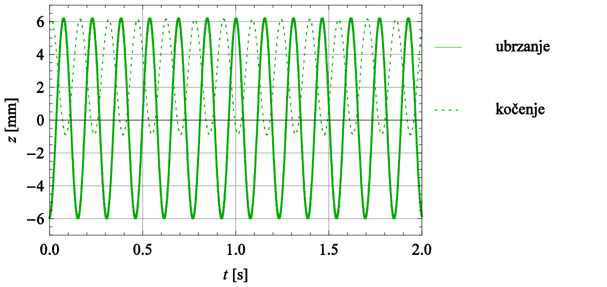
Sprovođenjem proračuna na osnovu teorijske analize izložene u [4], dobijene su dinamičke karakteristike nestacionarnih režima kretanja usled rada mehanizma za dizanje.

Na slici 12 je dat uporedni prikaz dinamičkih pomeranja u oba nestacionarna režima kretanja. Maksimalni dinamički ugib konstrukcije na mestu papuča konzola, za slučaj ubrzanja, iznosi 6.18 [mm], dok za slučaj kočenja, iznosi 6.14 [mm].

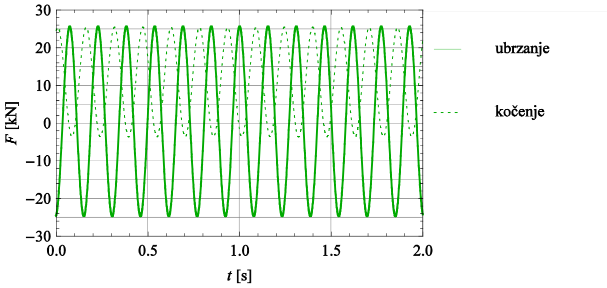
Na slici 13 je dat uporedni prikaz dinamičkih sila u oba nestacionarna režima kretanja. Maksimalna dinamička sila konstrukcije na mestu papuča konzola, za slučaj ubrzanja, iznosi 25.6 [kN], dok za slučaj kočenja, iznosi 25.4 [kN].

Određeni su odgovarajući dinamički koeficijenti. Dinamički koeficijent pomeranja konstrukcije na mestu papuča konzola, dobijen je deljenjem maksimalnog dinamičkog ugiba od 6.18 [mm], dobijenog na osnovu ekvivalentnog dinamičkog modela u režimu ubrzanja, sa maksimalnim statičkim ugibom od 5.92 [mm], dobijenim iz konačno-elementnog modela u programu ANSYS. Vrednost dinamičkog koeficijenta pomeranja od 1.044 govori da je došlo do apsolutnog maksimalnog povećanja ugiba za 4.4 [%] u režimu ubrzanja u odnosu kada se motorno vozilo drži na određenoj visini.

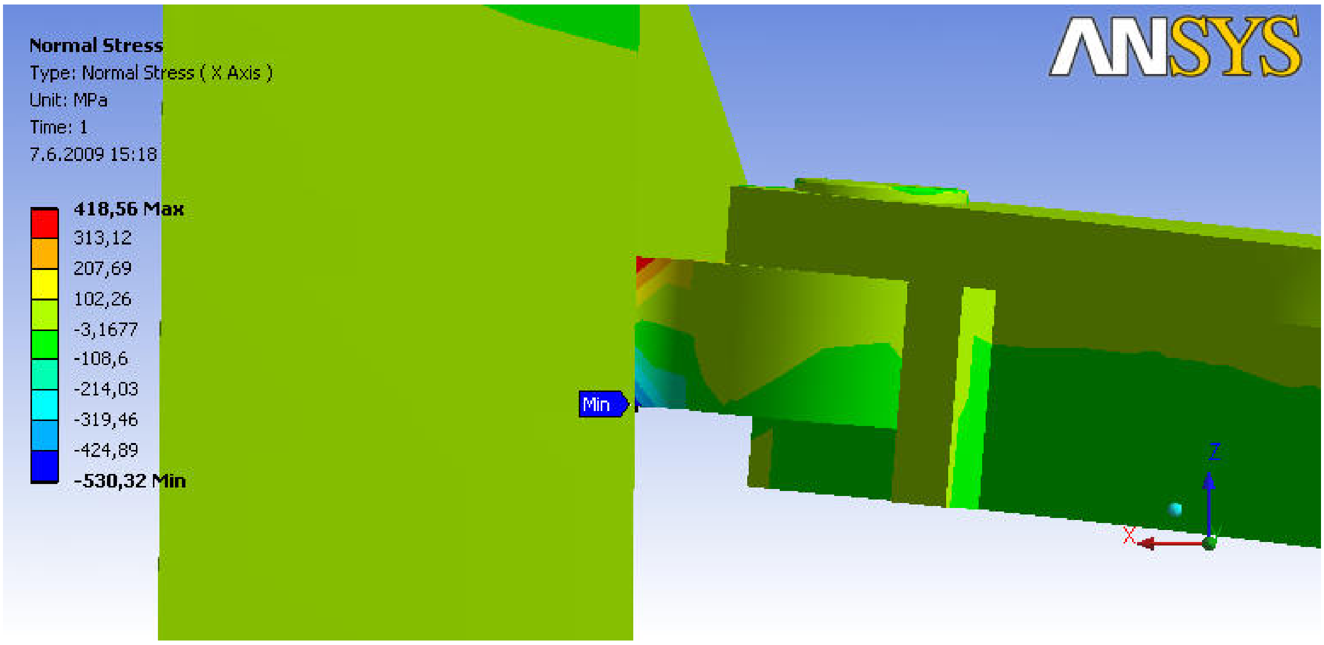
Analogno dinamičkom koeficijentu pomeranja, određen je i dinamički koeficijent sile elastične veze konstrukcije na mestu papuča konzola.



Slika 12: Zavisnost dinamičkog pomeranja od vremena



Slika 13: Zavisnost dinamičke sile od vremena



Slika 14: Detalj raspodele dinamičkih napona u zoni maksimalnih napona za konstrukciju dizalice

Dalje, maksimalna vrednost dinamičkog pomeranja u periodu ubrzanja pri dizanju motornog vozila, upotrebljena je u ANSYS-u za dobijanje maksimalnog dinamičkog napona u konstrukciji dizalice. Sprovođenjem statičke analize konačno-elementnog modela dizalice, za srednji položaj pokretnih nosača konzola i maksimalni dinamički ugib papuča konzola u režimu ubrzanja, dobijena je najveća vrednost napona na pokretnom nosaču konzola, slika 14.

Dinamički koeficijent napona konstrukcije, dobijen je deljenjem maksimalnog dinamičkog normalnog napona u periodu režima ubrzanja (530.32 [MPa]), slika 14, sa statičkim normalnim naponom (508.38 [MPa]), slika 8. Vrednost dinamičkog koeficijenta napona govori da je došlo do apsolutnog maksimalnog povećanja napona za 4.3 [%] u režimu ubrzanja u odnosu kada se motorno vozilo drži na određenoj visini.

Vrednost napona u istom preseku pokretnog nosača konzola, dobijena proračunom na osnovu važećih propisa i nacionalnog standarada SRPS, iznosi 584.63 [MPa]. Prema SRPS standardu uzima se u obzir propisana vrednost dinamičkog koeficijenta 1.15 [12].

Dinamički napon dobijen na osnovu dinamičkog pomeranja u režimu ubrzanja pri dizanju motornog vozila je niži od dinamičkog napona dobijenog primenom dinamičkog koeficijenta prema SRPS standardu za 10.2 [%].

4. ZAKLJUČAK

Ovim radom se pokušao dati određeni doprinos u oblasti dizaličnog inženjerstva. U prvom koraku je formiran konačno-elementni model za konkretan primer mehaničke dvostubne dizalice. Obezbeđeni su dobri kontaktni uslovi, sa jedne strane, između delova dizalice i sa druge strane, između papuča konzola i tereta. U toku simulacije se nisu pojavili loši elementi i sukobi. Pokazano je da je konačno-elementni model veoma važan i potreban za statičku i dinamičku analizu ovog tipa dizalice, usled postojanja složene interakcije mehanizma za dizanje i noseće strukture.

Analizom statičkih i dinamičkih karakteristika konstrukcije u programu ANSYS, pokazano je da konstrukcija MDD ima najveću statičku i dinamičku osetljivost kada se pokretni nosači konzola nalaze u srednjem položaju.

Utvrđeno je analizom nestacionarnih režima kretanja usled rada mehanizma za dizanje da je sa aspekta maksimalnih dinamičkih naprezanja režim ubrzanja kritičniji od režima kočenja. Pokazano je da se odgovarajući dinamički koeficijenti nalaze u okviru preporučenih vrednosti prema SRPS stanadardu.

Na kraju, rad ostavlja prostor za:

- optimizaciju noseće konstrukcije,
- analizu uticaja lokalnog savijanja, i
- analizu uticaja pomičnog opterećenja.

LITERATURA

[1] D. Ostrić i S. Tošić, "Dizalice", Mašinski fakultet, Beograd (Srbija), (2005)

[2] D. Ostrić, "Dinamika mosnih dizalica", Mašinski fakultet, Beograd (Srbija), (1988)

[3] http://www.wertherint.com

[4] R. Vasiljević, "Higher Levels Modelling Based on Inventor Software" Journal of Applied Engineering Science, Vol. 8(4), pp. 215–221, (2010)

[5] R. Vasiljević, Z. Petković and S. Bošnjak, "Applying Finite Element Method for Research Static and Dynamic Attribute of Electro-mechanical Two Post Lift", Proceedings of VII International Conference "Heavy Machinery HM 2011", Vrnjačka Banja (Serbia), 29 June-2 July 2011, pp. 99-104, (2011)

[6] D. Letić, "CAD mašinskiih elemenata i konstrukcija", Kompjuter biblioteka, Čačak (Srbija), (2004)

[7] ANSYS Inc., "ANSYS Structural Analysis Guide, Release 11.0", Cononsburg PA, (2007)

[8] Univerzal, "Uputstvo za rad i održavanje mehaničke autodizalice dvostubne DB2", Banja Luka, (1989)

[9] A. Obradović i S. Marković, "Zbirka zadataka iz teorije oscilacija", Narodna knjiga, Beograd (Srbija), (1996)

[10] V. Miltenović, "Mašinski elementi", Mašinski fakultet, Niš (Srbija), (1999)

[11] R. Čukić i D. Ružić, "Otpornost materijala 1", Mašinski fakultet, Beograd (Srbija), (1992)

[12] Z. Petković i D. Ostrić, "Metalne konstrukcije u mašinogradnji 1", Mašinski fakultet, Beograd (Srbija), (1996)

OZNAKE

z	generalisana koordinata koja opisuje oscilovanje motornog vozila, odnosno papuča konzola dizalice
m	masa translatornih delova svedena na mesto motornog vozila
c	krutost konstrukcije na mestu papuča konzola
Δ	ugib navojnog vretena
F_p	pogonska sila elektromotora svedena na noseće navrtke pogonskog i pratećeg navojnog vretena, odnosno na pravac dizanja motornog vozila
F_k	kočiona sila elektromotora svedena na noseće navrtke pogonskog i pratećeg navojnog vretena, odnosno na pravac spuštanja motornog vozila
F_w	statički otpor kretanju po jednom nosećem stubu

IMK-14 – Research & Development in Heavy Machinery 24(2018)3, EN85-91
UDC 621 ISSN 0354-6829

Dynamic Analysis of Mechanical Two Post Lift Using Combined Finite Element (ANSYS) and Analytical Method

Rade Vasiljević[1*]
[1]Kosančićeva 1A/4, 31109 Užice (Serbia)

In this paper discussed dynamic analysis of mechanical two post lift using combined finite element and analytical method. The first part of the paper is formed and solved finite element model of considered lift. Here is presented the application of a finite element method for determination of the state of stress deformation of construction mechanical two post lift presenting. The ANSYS program package is used. The second part of the paper are simplifying the finite element model of crane formed and solved her dynamic models. Dynamic characteristics of non-stationary regimes of moving because of the lifting mechanism have been obtained.

Keywords: Mechanical two post lift, Dynamic analysis, CAD/CAE, ANSYS, Analytical method

1. INTRODUCTION

Today, lifts should meet the spectra of items of design, in which important place is dynamic characteristics of lift. Dynamic analysis of lifts is very important, because it can clarify the real state of the loaded structure of lift [1], [2]. Other words, dynamic analysis of lifts it can clarify the design of the crane precisely determine the state of stress and the deformations of the elements mechanisms and the support structures. The analysis of dynamic characteristics of the lift brings us closer to the real stress state of the elements of lift, which enables obtaining of the optimal construction of the lift.

During studying of static and dynamic behaviour of mechanical two post lift (Figure 1) [3], complex interaction of the supporting columns and the lifting mechanisms present a special problem (Figure 2). Stiffness of driving and following column is equal to the superposition of supporting columns and threaded screws stiffness, whereat columns are console connected to the base of the crane, while threaded screw are beam placed in columns over bearings. Analytical method of determining the influential coefficients is complex, therefore the first step in static and dynamic analysis of mechanical two post lifts should not be creation of analytical models.

Nowadays, modern designing and construct of mechanical engineering is unthinkable without the applying of a computer [4], [5], [6]. From one side, 3D CAD became a standard. On the other side, making geometry of mechanical two post lift does not take much time and it is payable due to this crane presents the series product. Also, created geometry in 3D CAD software can be used in static and dynamic analysis. This means that the first step in static and dynamic analysis of mechanical two post lift should be making 3D models in specialized software package for 3D modelling, and then determining the static and dynamic parameters in a specialized software package for finite elements. Finite element software package for finite elements. Finite element method (FEM) belongs to the group of modern numerical methods [5]. It can be used for static and dynamic analysis of complex parts and assemblies [4], [6], [7].

The mechanical two post lift of a type DB2 ("Univerzal" - Banja Luka) is selected for the object [8].

Figure 1: Typical mechanical two post lift [3]

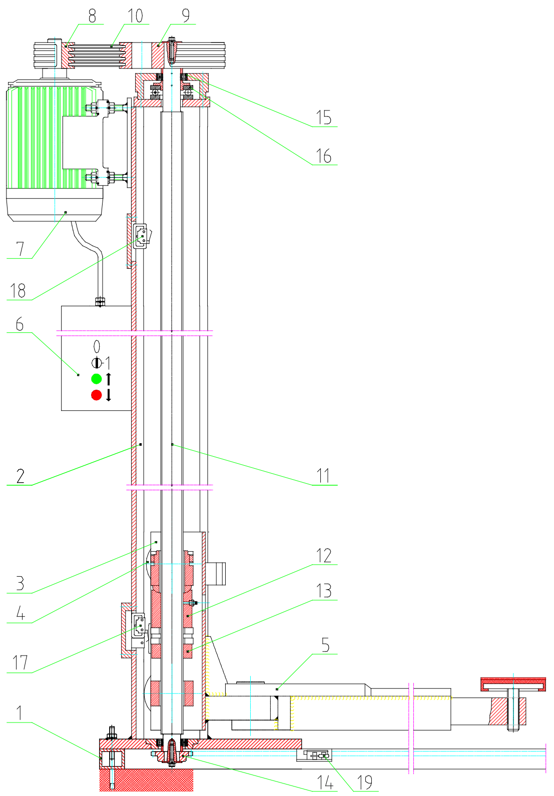


Figure 2: Driving column of mechanical two post lift

*Corresponding author: Kosančićeva 1A/4, 31109 Užice, email:r.r.vasiljevic@gmail.com

2. THE FINITE ELEMENT ANALYSIS - FEA (ANSYS)

Structural diagram of solving finite element analysis of mechanical two post lift in the ANSYS program is represented at Figure 2. The analysis of the presented structural diagram is given in Ref. [5].

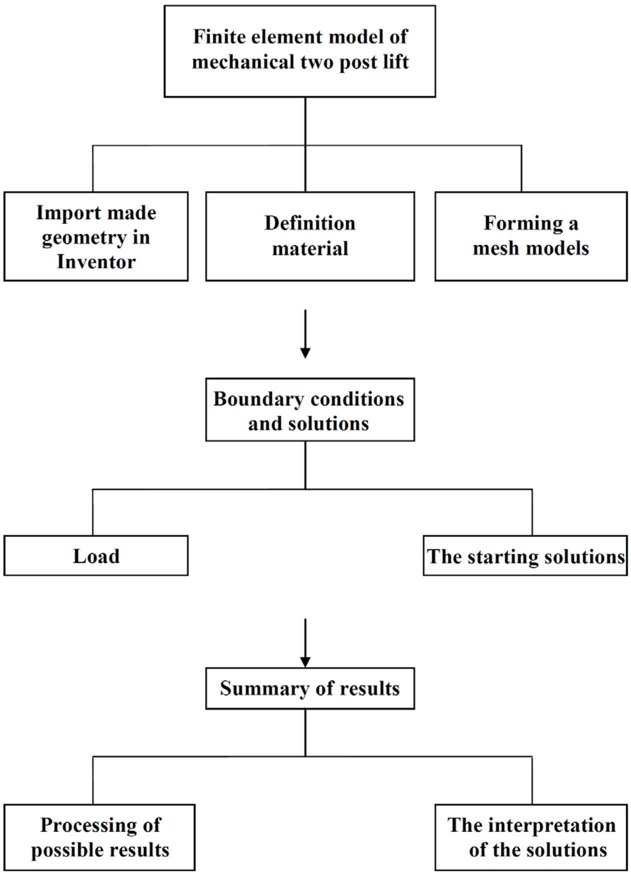


Figure 2: Structural diagram of FEA in ANSYS

2.1. Formation of finite element model

Original 3D solid model of implemented solution for mechanical two post lift of type DB2, by manufacturer "Universal" from Banja Luka [8], applying specialized software package Autodesk Inventor, Figure 3, is made (CAD environment). Technology of making this 3D model is exposed in Ref. [5].

For purposes of determining the static and the dynamic characteristics, original 3D solid model, made in Autodesk Inventor, has been imported into specialized software package ANSYS, Figure 4.

At the created 3D model, according to the manufacturer recommendation, the base of crane is connected for the concrete plate dimensions 3500×1600×180 [mm] over the anchor bolts.

At this solution regarding 3D model of mechanical two post lift, threaded screws are radial and axial fixed in upper case, while at the lower case they are radial fixed and axial free.

Crane load is represented by homogeneous steel plate, mass $m_Q = 2500$ [kg].

In accordance to the created geometry of mechanical two post lift, mesh finite element generation in ANSYS program, Figure 5.

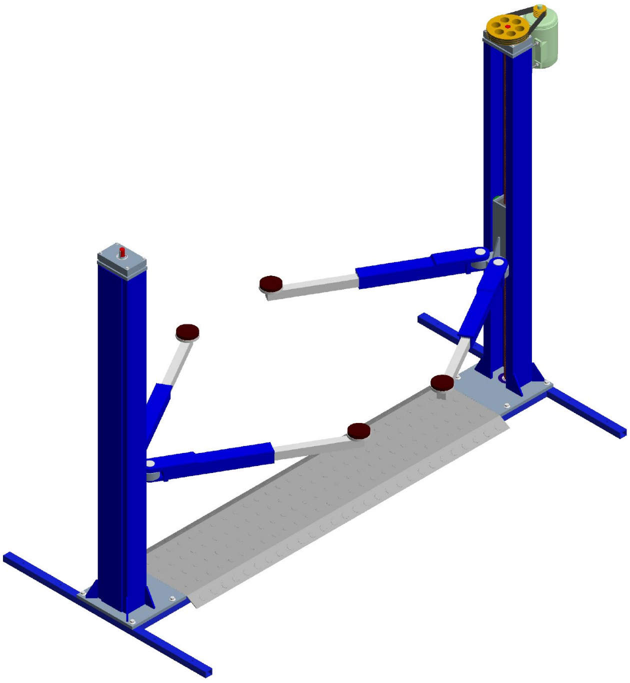


Figure 3: 3D model of mechanical two post lift - Inventor

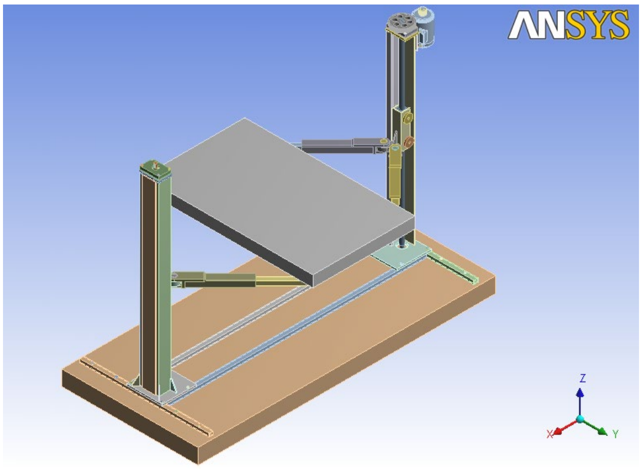


Figure 4: 3D model of mechanical two post lift - ANSYS

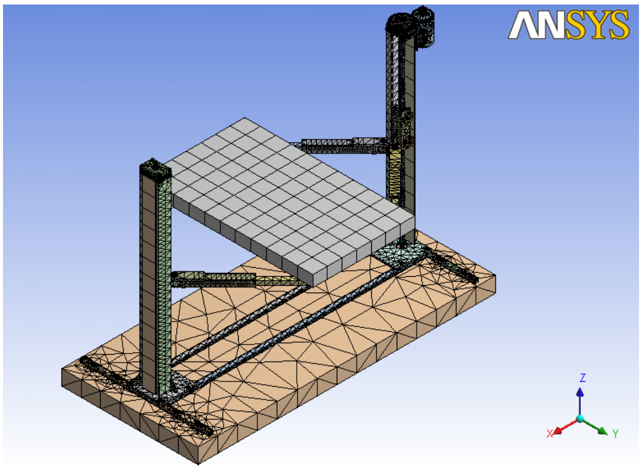


Figure 5: Mesh FE generation of 3D model of mechanical two post lift - ANSYS

The finite element analysis has been done at the formed FE model using the software ANSYS (CAE environment) [7]. Three characteristic positions of mobile console carriers are analysed:

- I - lower end position of mobile console carriers,
- II - middle position of mobile console carriers, and
- III - upper end position of mobile console carriers.

2.2. Static structural analysis

Supporting structure as well as threaded screw character and maximum values of the following static characteristics, have been analysed:

- static displacements, and
- static stresses.

2.2.1. Static displacements

By comparative analysis of displacements in regard with three characteristic cases, it was found that maximum displacements are at console pedals zone.

Displacements of console pedals are slightly larger in the upper end position according to middle position of mobile consol carriers ($\delta_{cp,II}$ =5.92 [mm], $\delta_{cp,III}$=6.04 [mm]).

Displacements of the columns tips are the largest in middle position of the mobile console carriers and specially at foolowing columns ($\delta_{ct,II}$=0.78 [mm]).

Threaded screws, according to the change of stiffness, have the largest displacement in the middle position of mobile console carrier ($\delta_{ts,II}$ =0.295 [mm]).

Comparing the values of displacements for all three positions can be concluded that crane is the most sensitive to static effects when mobile console carriers are located in the middle position (II). It means that from the aspect of static displacement, middle position of mobile console carriers is critical position.

Deformation of crane when it is loaded with maximum weight of ballast at upper end position are represented at Figures 6 and 7.

2.2.2. Static stresses

Analogically to the analysis of displacement, from the comparative analysis of normal stresses for three characteristic cases, it is noticeable that maximum normal stresses are in the zone of mobile console carrier.

Normal stresses in the zone of mobile console carrier are slightly larger in upper end position than in the middle position of mobile console carriers ($\sigma_{mc,II}$=508.38 [MPa], $\sigma_{mc,III}$=540.83 [MPa]).

The threaded screws, according to the change of stiffness and derived stresses distribution, have concentration of larger normal stresses in the middle position of mobile console carrier. Consequently, supporting columns have concentration of larger normal stresses in the middle position of mobile console carriers.

Also, based on the distribution of stresses in the interaction zone of leading wheels of the mobile console carriers and supporting columns, local bending is noticeable and it is larger for the middle position of mobile console carriers.

The detail of the distribution of static stresses in the zone of maximum stresses for crane construction, when it is loaded with maximum weight ballast in the middle position, is represented at Figure 8.

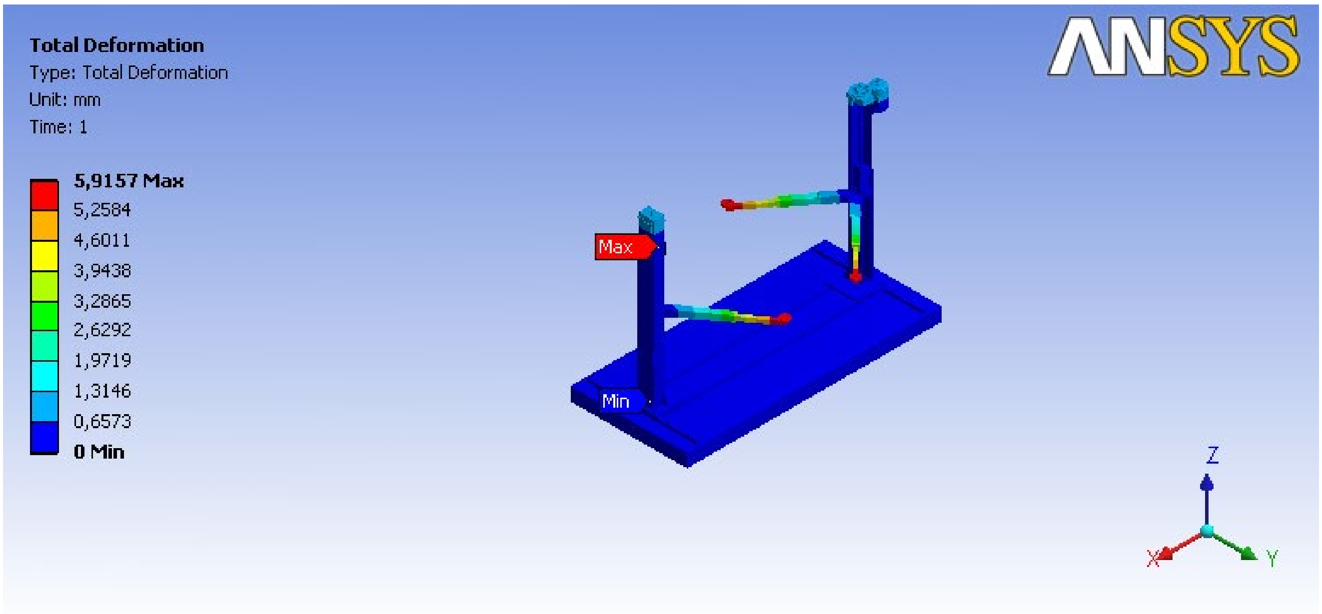


Figure 6: Total deformation of mechanical two post lift

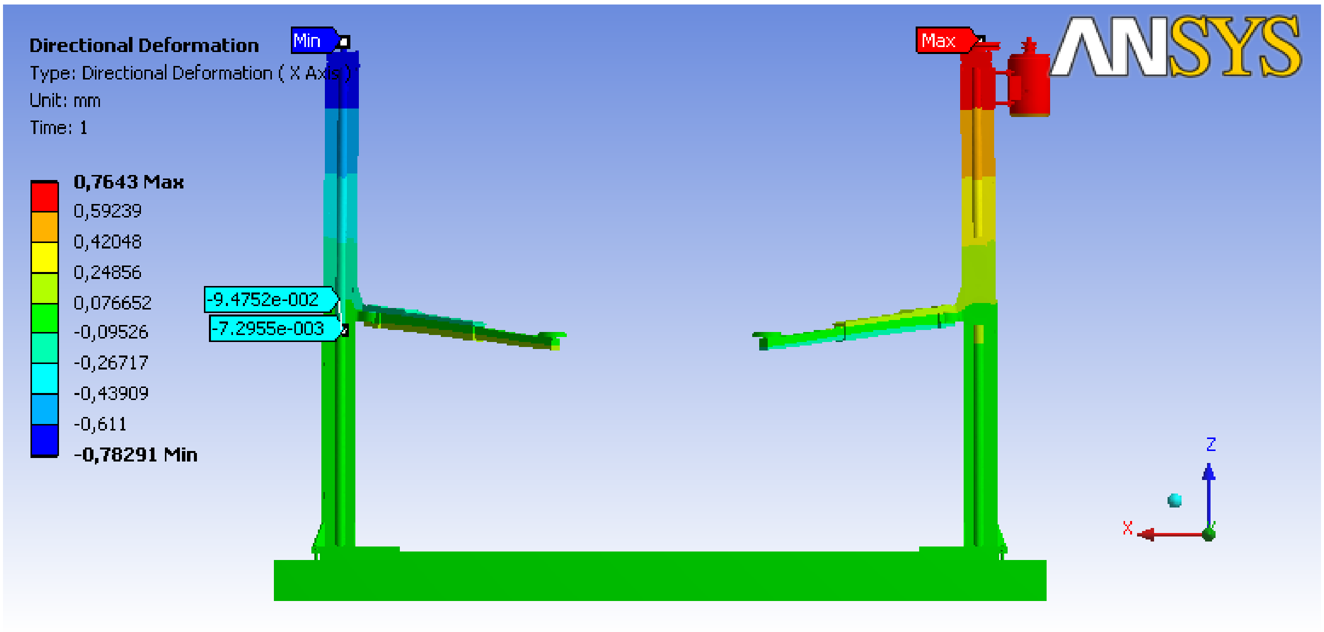


Figure 7: Deformation in the direction of X of mechanical two post lift

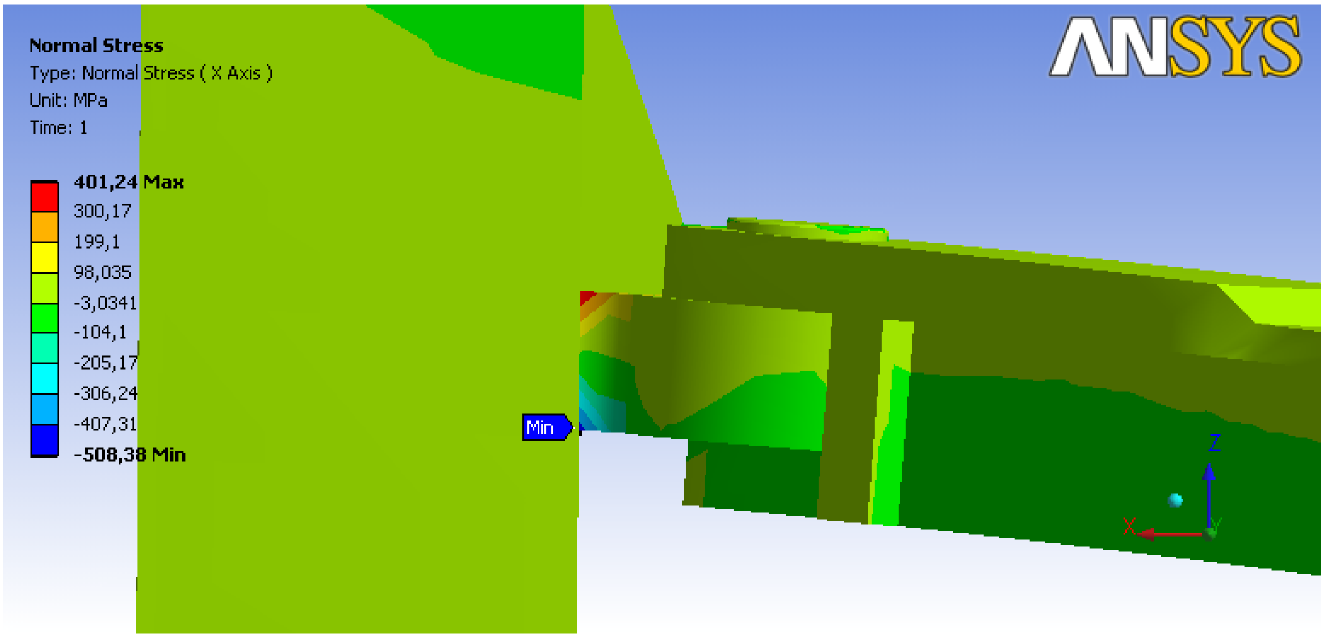


Figure 8: Detail of distribution of static stresses in zone of maximum stresses for crane construction

2.3. Modal analysis

Efficient and reliable work of mechanical two post lift is under effects of its own frequencies. Own frequencies are directly depending on the stiffness and mass of the construction. Determination these own frequencies is the first step in implementing of dynamic analysis of mechanical two post lift.

Numerical determination of the modal parameters are conducted by finite element method, by using software package ANSYS [7]. The following modal parameters are analysed:

- frequencies of oscillation, and
- forms of oscillation.

2.3.1. Frequencies of oscillation

For all three positions of mobile console carriers, first six oscillation frequencies of 3D model are determined, which is default number in the ANSYS.

Comparing of the frequencies values for all three cases, it can be concluded that crane is the most sensitive dynamic from the aspect of the first lowest (basic) frequency in the case when mobile console carriers are in upper end position. But, as fundamental frequency in upper end position is slightly different from fundamental frequency in the middle position, from one side, and as stiffness of threaded screw is the lowest in the middle position and displacement of the crane columns ends is the largest in the middle position, on the other side, so position II is the most proper for further description of dynamic state of the construction.

The first six frequencies, when the crane is loaded with maximum weight of ballast in the middle position, derived by applying software package ANSYS, shown in Table 1.

Table 1: Frequencies

Mode	Freq., f [Hz]	Cir. freq., ω [s^{-1}]	Period, T [s]
1	6.55	41.14	0.153
2	8.25	51.83	0.121
3	11.12	69.85	0.09
4	14.56	91.42	0.069
5	14.79	92.90	0.068
6	23.84	149.73	0.042

2.3.2. Forms of oscillation

Modal analysis gave first six oscillation forms of mechanical two post lift for the middle position of mobile console carriers, when the crane is loaded with maximum weight of ballast.

The first main form of oscillation is represented at Figure 9. At Figure 9 can be observed that form of oscillation in the direction of lifting and lowering the motor vehicle (1st mode) corresponds to the elastic line in the vertical plane, apropos to the deformed form due to retention of vehicle on the certain elevation head. Also, there is dominance of console oscillation in the vertical direction, relative to other elements of construction.

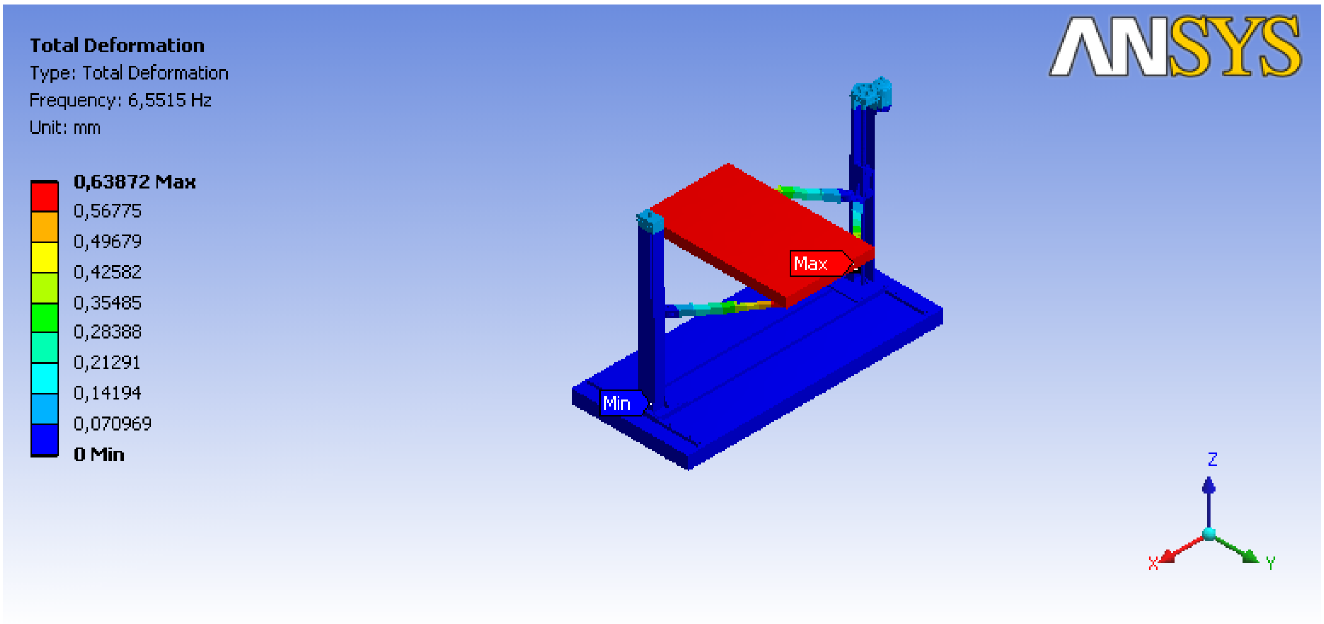


Figure 9: 1st Mode of mechanical two post lift, T_1=0.153 [s]

3. ANALITC ANALISIS OF NON-STATIONARY MOTIONS

The analysing of the dynamic characteristics of the lift in non-stationary regimes, due to work of lifting mechanism was conducted on the basis of finite element analysis.

By simplification of 3D models of lifts, analytical dynamic models were formed, Figures 10 and 11, so these to retain the same basic supporting structure of the oscillation frequency in vertical direction. For the formation of equivalent mathematical models as the basis used static and dynamic parameters obtained by FEA in ANSYS. The creation of this models was preceded by the review of Refs. [1], [2], [9], [10], [11].

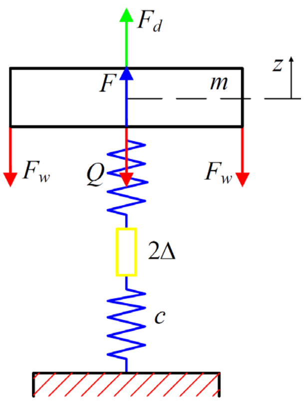


Figure 10: Dynamic model of mechanical two post lift mode acceleration in weightlifting

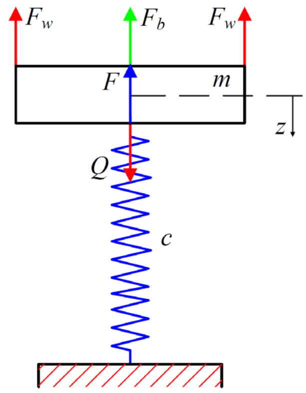


Figure 11: Dynamic model of mechanical two post lift mode braking in descending

Differential equations of dynamic models that describe the dynamic behaviour discussed lifts in a vertical plane in non-stationary regimes because of the mechanism for lifting the vehicle, read:

$$m\cdot\ddot{z}+c\cdot z=F_d-m\cdot g-2F_w \tag{1}$$

$$m\cdot\ddot{z}+c\cdot z=m\cdot g-F_b-2F_w \tag{2}$$

Conducting calculations on the basis of theoretical analysis exposed in [4], were obtained dynamic characteristics of non-stationary regimes of movement because of the lifting mechanism.

In Figure 12 gives a comparison of the dynamic deformation in both non-stationary regimes of movement. Maximum dynamic deformation construction in the place of shoe console, in case of acceleration, is 6.18 [mm], while in case of braking, is 6.14 [mm].

In Figure 13 presents a comparison of the dynamic force in both non-stationary regime of movement. Maximum dynamic force structure in the place of shoe console, in case of acceleration, is 25.6 [kN], while in case of braking, is 25.4 [kN].

The appropriate dynamic coefficients are determined. Dynamic coefficient of moving structures in the area of shoe console, was obtained by dividing the maximum dynamic deflection of 6.18 [mm], which is obtained on the basis of the equivalent dynamic model in the regime of acceleration, with the maximum static deflection of 5.92 [mm], obtained from finite element model in ANSYS program. The value of dynamic coefficient of deformation of 1.044 suggests that there was an absolute maximum increase in the deformation of 4.4 [%] in the regime of acceleration in relation to when the vehicle was held at certain height.

Analogically the dynamic deformation determined by the ratio of force and dynamic coefficient of elastic connections structure in place shoe console.

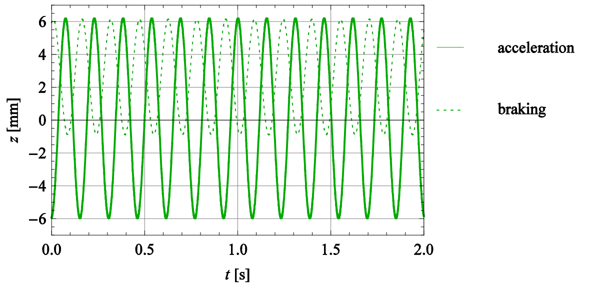


Figure 12: Dependence of dynamic displacement of time

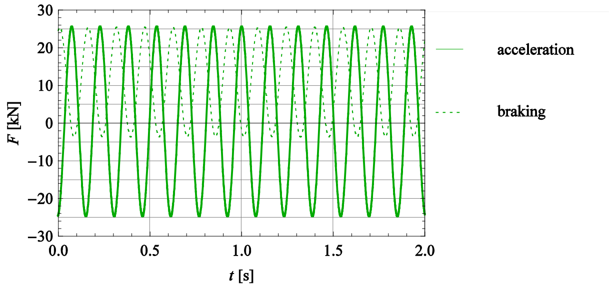


Figure 13: Dependence of dynamic force of time

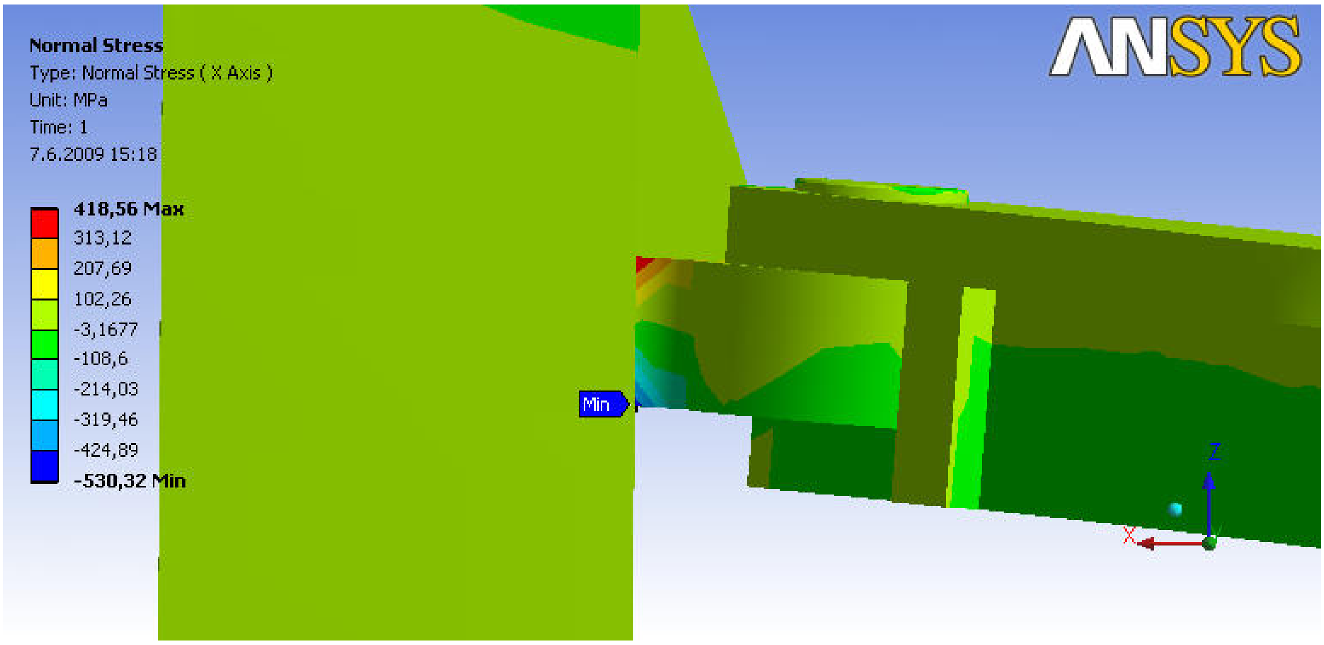


Figure 14: Detail of distribution of dynamic stresses in zone of maximum stresses for crane construction

Further, the maximum value of dynamic deformation in the period of acceleration of the lifting of the motor vehicle was used in ANSYS to obtain the maximum dynamic stress in the lift construction. The implementation of static analysis finite element model lift, the central position of mobile carrier console, and the maximum dynamic deformation shoe console mode acceleration, the resulting stress is the largest on the mobile carrier console, the Figure 14.

Dynamic coefficient of stress design, was obtained by dividing the maximum dynamic normal stress during the regime of acceleration (530.32 [MPa]), Figure 14, with normal static stress (508.38 [MPa]), Figure 8. The value of dynamic coefficient of stress shows that there was an absolute maximum stress increase of 4.3 [%] in the regime of acceleration in relation to when the vehicle was held at a certain height.

Stress at the same intersection of mobile carrier console, as result of calculations based on current regulations and national standards SRPS, is 584.63 [MPa]. The SRPS standard takes into account the prescribed value of dynamic coefficient of 1.15 [12].

Dynamic stress obtained on the basis of dynamic deformation in the regime of acceleration by lifting vehicle is lower than the dynamic stress obtained by applying the dynamic coefficient of SRPS standard for 10.2 [%].

4. CONCLUSION

This paper tried to give a contribution in the field of lift engineering. In the first step, the FE model for concrete example of mechanical two post lift is formed. Good contact conditions are provided, from one side, between parts of the crane and from the other side, between console pedals and the ballast. During simulation there was no appearance of bad elements and conflicts. It is approved that FE model is very important and necessary for the static and dynamic analysis of this kind of crane, due to the existence of a complex interaction of mechanism for lifting and supporting structure.

The analysis of static and dynamic characteristics of structures in ANSYS has been shown that construction electro-mechanical two post lifts has the highest static and dynamic sensitivity when mobile carriers console are located in the middle position.

It was affirmed by analysis of non-stationary regimes due to movement of the lifting mechanism that the point of maximum dynamic strain regime of acceleration more critical braking regime. It was shown that the corresponding dynamic coefficients are within the recommended values of the SRPS standard.

Finally, the paper leaves space for:

- optimization of supporting structure,
- analysing of the influence of the local bending, and
- analysing of the influence of the moving load.

REFERENCES

[1] D. Ostrić i S. Tošić, "Dizalice", Mašinski fakultet, Beograd (Srbija), (2005)

[2] D. Ostrić, "Dinamika mosnih dizalica", Mašinski fakultet, Beograd (Srbija), (1988)

[3] http://www.wertherint.com

[4] R. Vasiljević, "Higher Levels Modeling Based on Inventor Software" Journal of Applied Engineering Science, Vol. 8(4), pp. 215–221, (2010)

[5] R. Vasiljević, Z. Petković and S. Bošnjak, "Applying Finite Element Method for Research Static and Dynamic Attribute of Electro-mechanical Two Post Lift", Proceedings of VII International Conference "Heavy Machinery HM 2011", Vrnjačka Banja (Serbia), 29 June-2 July 2011, pp. 99-104, (2011)

[6] D. Letić, "CAD mašinskiih elemenata i konstrukcija", Kompjuter biblioteka, Čačak (Srbija), (2004)

[7] ANSYS Inc., "ANSYS Structural Analysis Guide, Release 11.0", Cononsburg PA, (2007)

[8] Univerzal, "Uputstvo za rad i održavanje mehaničke autodizalice dvostubne DB2", Banja Luka, (1989)

[9] A. Obradović i S. Marković, "Zbirka zadataka iz teorije oscilacija", Narodna knjiga, Beograd (Srbija), (1996)

[10] V. Miltenović, "Mašinski elementi", Mašinski fakultet, Niš (Srbija), (1999)

[11] R. Čukić i D. Ružić, "Otpornost materijala 1", Mašinski fakultet, Beograd (Srbija), (1992)

[12] Z. Petković i D. Ostrić, "Metalne konstrukcije u mašinogradnji 1", Mašinski fakultet, Beograd (Srbija), (1996)

NOMENCLATURE

z generalized coordinates describing the oscillation of the vehicle or the shoe console lift

m mass translational parts reduced to the position of the motor vehicle

c construction stiffness in place shoe console

Δ deflection threaded spindle

F_d electric motors driving force is reduced to a supporting nut driving and accompanying threaded spindle or the direction of lifting of the motor vehicle

F_b electric motors braking force is reduced to a supporting nut driving and accompanying threaded spindle or the direction of lowering of the motor vehicle

F_w static resistance motion on single support column