https://erjm.journals.ekb.eg
ISSN: 1110-1180

DOI: ---------

ERJ
Engineering Research Journal
Faculty of Engineering
Menoufia University

Assessment of Parallel Robot Dynamic Characteristics Using Experimental Modal Analysis and Finite Elements

Ahmed A. Deabs[1], Fawkia R. Gomaa[1], Khaled M. Khader[1,2]

[1] Production and Mechanical Design Engineering Department, Faculty of Engineering, Menoufia University, Shebine El-kom, Egypt;
[2] Currently on leave to Delta Technological University, Industrial Zone, Quessna, Menoufia, Egypt
(Corresponding author: ahmed_deabs@sh-eng.menofia.edu.eg)

ABSTRACT
The high accuracy of industrial robots is the main aim of designers and manufacturers, one of the effective factors to get the goal is the stability of the robot structure, the determination of the structural dynamic characteristics is the main step to evaluate the performance also unlock the knowledge of amendment and improvement of the structure to get the optimum design. In this paper, two methods were applied to evaluate the dynamic structural performance of three degrees of freedom parallel robot, firstly, experimental modal analysis was applied to a multi-model with different platform dimensions using a data acquisition system, the natural frequencies, and damping ratios for all models were obtained to be evaluated and correlated with the second method. The measured models were modeled using Solidworks software and exported to Ansys finite element (FE) software, the modeled systems were used to obtain natural frequencies, damping ratios, and mode shapes from frequency response curve (FRF) and modal analysis, the results of experimental and FEM work were correlated to evaluate the system performance and verify the accuracy of the two methods. The results give a clear view to operators about the range of frequencies that must be avoided during the selection of machining operation, provide the scope of errors between the used methods, and supply a valuable guide to evaluate the quality of the structural integrity of the parallel robot.

Keywords: *Modal Analysis; Frequency Response Function; Dynamic Characteristics; Parallel Robots; Machine Tools*

1. Introduction

The industrial robot's performance depends on more than one factor, one of the main factors is the dynamic characteristic (natural frequencies, modes shapes, and damping factor) of the robot structure. There is more method to determine the robot's dynamic characteristic, such as experimental modal analysis (EMA) and Finite Element Method (FEM) using engineering software such as Ansys, Adams, Solidworks, Abaqus, ProEngineer, …. etc. Identifying dynamic properties is very useful in machine structural performance evaluation, modification, and machining parameters selections.

For experimental measurements, more research was performed on more than one robot type, PUMA 560 robot was analyzed using experimental modal analysis to identify the natural frequencies at the static and power-on state and four different configurations., the results were compared with previous research study, the results give acceptable correlations [1]. Elosegui [2] described the nonlinear characteristic of the Puma 560 robot by using EMA, two models were proposed and measured to get the system response, the results were compared to identify the nonlinearities and select suitable measurement techniques.

The dynamic structural evaluation of the redesigned LOLA walking robot was obtained by performing EMA on the structure at more than one point, the results were also compared with open-loop transfer functions results [3], Wu and Kuhlenkoetter [4] obtained the dynamic stiffness of IRB 4400 industrial robot by using experimental modal analysis, the results from measurements were converted by two mathematical approaches to get the dynamic stiffness, the comparison showed the significant difference between them and declared the best method. Modal parameters (natural frequencies, modal damping ratios, and mode shapes) of ABB IRB 6660 robot were identified using EMA, the results were discussed to improve the robot structure, reduce the vibration and increase the accuracy [5].

This paper was presented in the 1st Inter. Conf. in Technological University Education and its Role in Industry, Energy and Environmental Conservation (Cairo 15-16 May, 2022) Upon arrangement with Delta Technical University.

A Six-Axis Industrial Robot natural frequencies were described by using EMA, also two methods depending on the design of experiments were applied to predict the system's natural frequencies with the presence of joint configuration, the results were compared and achieved a good correlation [6], Guodong, Junchuan and Lili [7] studied the behavior of 6-DOFs Series Robot using modal testing to solve the complexity of theoretical modeling, the modes shapes and natural frequencies were investigated, and author recommended that the joints must be stiffened and reasoned frequencies must be avoided.

For finite element analysis, different types of robots were analyzed to get the dynamic characteristics and improve the robot performance, the effect of robot arm cross-section was survived using Ansys software, the results presented that the circular section gives high frequencies range than other section and hollow circular section gives less equivalent stress [8], A modal and harmonic analysis were performed to a robotic arm using Ansys software to improve the structure design, the natural frequencies were used to avoid resonance locations, the displacement obtained was used to make the structural improvement [9], the same analysis using the change of material structure was applied to articulated robotic arm at different research [10].

A Cable-driven parallel robots (CDPRs) vibration was investigated using FEM and modified analytical formulation, the methods results were used to correlate the FEM SAP2000 software results to improve the prediction of (CDPRs) vibration problem [11]. Do and Park [12] studied the effective frequency of cable in (CDPRs) using FEM, the results showed that high speed produces more effective frequencies than low speed also showed that frequency is affected with the position and tension in cables. The rigid FEM was used to simulate the (CDPRs) using damper, spring, and rigid bodies to solve the problem of cables flexural rigidity [13].

A gear system of 3-DOF Wrist Mechanism used in SCARA robot was analyzed using FEM software to reduce the weight, check the stability and select the suitable gears of the wrist [14], the comparison between to FEM methods was applied using a parallel robot to compare the body flexibility results, the first method depended on using model directly from CAD software to Ansys, the second method depended on converting the model to a flexible body using Adams software then export the model to Ansys, the results showed that the second method is more realistic and authentic [15, 16].

Dawood and Kavati [17] used Ansys to test the modified Industrial Omron Hornet 565 Delta robot with 3-axis using static analysis, also different loads and materials were used to check the performance, the results showed the design within the safe limit, A FEM modal analysis was applied to an industrial robot with eight-axis used in the painting system to clear the state of dynamic performance, the authors recommend that robot mist contain more light and stiff material, and joint modeling must be precision considered [18], Rueda and Ángel [19] investigated the flexible, dynamic performance of delta parallel industrial robot using Ansys and SolidEdge software, the analysis showed the ability to select a motor, material, and identification of allowable forces to get low power consumption, cost and high efficiency.

A 3-PPSS Parallel structural kinematic analysis was stated using Ansys software to determine the optimum dimension of the robot links and select the suitable motor according to stress analysis on the joints [20].

A 3-RPS Parallel Robot was simulated using Adams and Ansys software to evaluate the dynamics of the rigid-flexible coupling system, the results showed the high precision of the simulation also aid in structure design and optimization [21], Qinghua and Xianmin [22] studied the effect of the temperature changes of 3-RRR flexible parallel robots, the results showed using stress analysis the significant change can be produced after temperature variation, also clear the view of taking temperature into consideration during parallel robot performance analysis.

Multi-software (ProEngineer, Adams, and Ansys) platform was used to investigate the dynamic of 3 RPS parallel robot, the displacement difference error between each software didn't exceed 0.0002m with a period of 4.2 seconds, the used method improved the results of robot dynamics and help in next structural improvement [23], The fatigue problem of manoeuvring laparoscopic medical robot tips was analyzed using FEM software, the result investigated the allowable load and angles that permit to do long time surgical operation with safe limits [24].

The combination of experimental and FEM analysis was applied by some researchers, the Modal parameters of flexible robot arm joints and links were investigated using Ansys FEM software and EMA, the difference between the two methods' results were proposed due to the dissimilarity between the actual model joints, and CAD model [25], Li and Yang [26] studied the dynamic characteristic of 2-DOF translational Pick-and-place parallel robot with flexible Links using FEM and EMA, the obtained natural frequencies showed that the robot has a high ability to work at high speeds, The two models of mobile platform and 6 DOF articulated-arm robot were dynamically analyzed using EMA and multibody system FEM model, the natural frequencies and mode shapes were obtained and correlated, the maximum error percentage didn't exceed 15%, the results opened the door to select the suitable actuators [27].

Two models of robots (KUKA KR90 R3100 robotic arm) were modeled using multibody system FEM to predict the dynamic behavior, the results were correlated with experimental measurements, some mode shapes can't be determined due to lack of inertia knowledge of joints, but all results could be suitable to predict the dynamic characteristics of the models [28], A 6 PUS PKM was dynamically analyzed using EMA and theoretical modal analysis, the second method was applied using Adam FEM software to describe the dynamic behavior and mode shapes, the results were compared and summarized with a map [29].

3-DOF parallel kinematics manipulator dynamic characteristics were investigated using EMA and FEM software, the results differences showed that more accurate modeling must be taken into consideration. Also, EMA must be modified by changing the type of excitation [30], a welding robot with two laser beams was tested using orthogonal experiment design and FEM modal analysis at a different location and joints angles, the natural frequencies were obtained, and differences between values were discussed to consider this study as a reference of robot control and structure optimization [31], The table of 4-DOF parallel machine tool was modeled using Solidworks and solved using FEM Ansys software, also the system was measured using EMA, the results were correlated to identify the frequencies and speed must be avoided during machining, and the suitable height of the table was selected [32].

In this paper, EMA and FEM method using Ansys are used to determine the dynamic characteristics of 6 different models' parallel robot, also the results are correlated, the selection of models was dependent on changing the lower (Tool) platform dimension relative upper platform as shown in Table (1), therefore the used models are (20-20), (20-20), (25-20), (25-25), (30-30), (30-25), the influence of changing platform is studied according to the maximum location of deformation in the model components.

Table 1 – The dimension of the used model platform side length (cm).

Upper Platform **Lower Platform**	**20**	**25**	**30**
20	√	√	√
25	X	√	√
30	X	X	√

2. Analysis and Basic Theories

The degrees of freedom (DOF) of the parallel robot can be determined using the Grübler formula

$$DOF = \Sigma(Freedoms\ of\ bodies) \\ +No\ of\ independant\ constrains$$

$$DOF = \mathrm{m}(N-1-J) + \sum_{i=1}^{J} c_i \qquad (1)$$

$N =$ No. of bodies, including ground

$J =$ No. of Joints

$m = 3$ for planar, 6 for spatial bodies

$DOF = 6(8-1-9) + [(3X2) + (6X2)] = 6$

By applying the Grübler formula, the model has a 3 DOF for each platform, when the upper platform is fixed, the degrees of freedom will be 3 DOF.

The general equation of motion of n degrees of freedom is:

Where

$$[M]\{\ddot{X}\} + [C]\{\dot{X}\} + [K]\{X\} = \{F\} \qquad (2)$$

$[M]$ is the inertia matrix

$[C]$ is the damping matrix

$[K]$ is the stiffness matrix

$\{F\}$ is the force vector

$\{X\}$ Refers to the structure displacement.

In the harmonic analysis used in Ansys software, the equation forms used in the software are:

$$\{F\} = \left\{F_{max} e^{i\psi}\right\} e^{i\Omega t}$$

$$\{u\} = \left\{u_{max} e^{i\phi}\right\} e^{i\Omega t}$$

$$(-\Omega^2[M] + i\Omega[C] + [K])(\{u_1\} + i\{u_2\}) = (\{F_1\} + i\{F_2\}) \quad (3)$$

Where

ψ = Force phase shift

ϕ = Displacement phase shift

Ω = Imposed Circular Frequency

$u_1 = Real\ displacement\ vector$

$u_2 = Imaginary\ displacement\ vector$

$F_1 = Real\ force\ vector$

$F_2 = Imaginary\ force\ vector$

3. Experimental Modal Testing

The experimental modal testing was performed to identify the dynamic characteristics of the six different models, each model with hinged using a spring from the upper platform, as shown in figure (1), due to a lack of measurement software license, the accelerometer [Tri-axial accelerometer B&K (4506)] was mounted at two points with different direction to collect the data from 3 axis as shown in figure (2), each direction collects data from two axis, the results then collected and arranged to remove all repeated information.

The impact hammer B&K (8202) testing was used to make excitation to the model, the signals were obtained using a data acquisition type B&K (3160-A-042) analyzer which connected to PC, the signals were collected using B&K Pulse 17.1 software which was installed and configured to PC system and pre-configured.

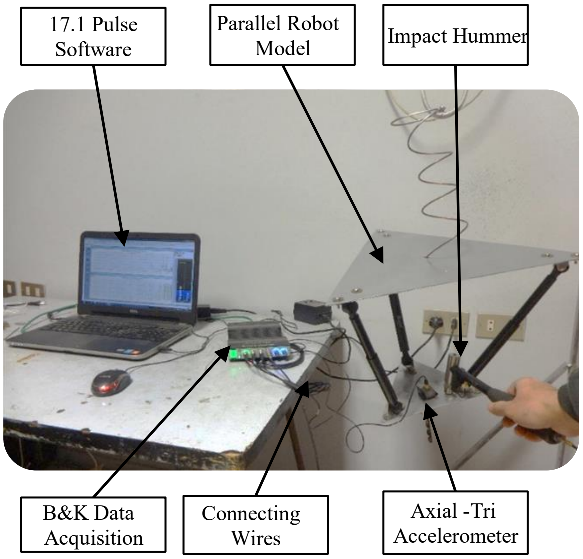

Figure 1 - Arrangement of the Experimental Modal Analysis Set-Up.

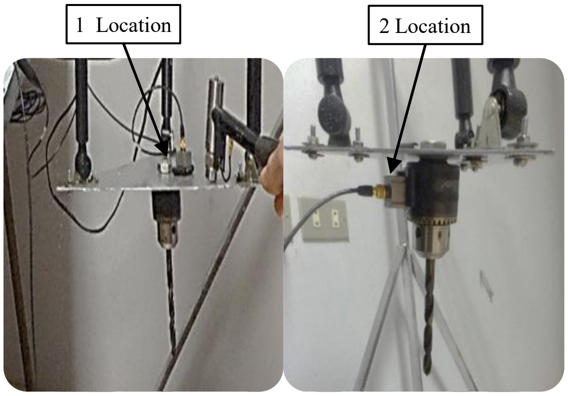

Figure 2 - Measurement Locations on the model.

4. Finite Element Modelling

Firstly, the system was modeled and assembled using Solidworks software with all components (83 Components), then it was exported to Parasolid (x_t) extension. Via Ansys software, the model Parasolid file was imported to be analyzed through the suitable analysis.

There is more type of analysis in Ansys, in this paper, harmonic analysis was used to get the frequency response curve (FRF) and mode shapes, the material damping ratio was taken into consideration for each party, by using the FRF curve, the system natural frequencies and maximum deformation location were obtained for all models.

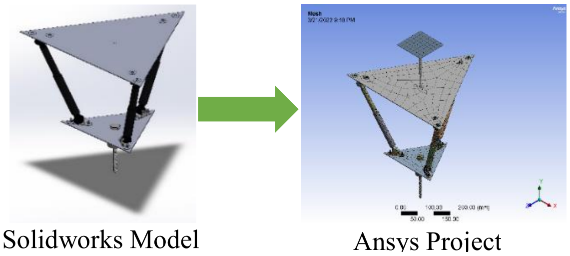
Solidworks Model
Ansys Project

Figure 3 - Exportation Model from Solidworks to Ansys

5. Results & Discussion

The correlation between EMA and FEM results is a powerful tool to get the accuracy of each method and identify the maximum error, Table (2) shows the comparison between natural frequencies of EMA and FEM, also error ratio between results, the maximum average error ratio is 6.74% at the (30-20) model, the minimum average error ratio is 3.46% at the (25-20) model, the table present that some results of EMA is missed with relative to FEM, the reasons for this dissipated mode frequency are the high damping of the material and the lack of Pulse software license, on the whole, the ratio of lost frequencies didn't exceed 8% of each model frequencies.

Table (3) display the deformation value and location of each model at all frequencies also, the average deformation rate of each model was indicated, the maximum value of average deformation is 1.727406mm at the (25-25) model, the minimum value is 0.002705mm at (25-20) model, table (4) shows the maximum deformation location in each component for all models, the joints and links take the maximum percent of deformation, the ultimate link deformation always occurs when the length of the platforms ratio becomes greater as referred in the model (25-20) and (30-20), the maximum joints deformation happened in all cases when the upper and lower platform have the equal length, the platform higher deformation occurs only one time at (30-25) model.

Figure (5) present a sample of the EMA FRF and coherence curves for all models with two measurement locations for one model, the coherence curve indicates the status of the measurement and select the range of perfect frequencies in FRF, in general, the coherence curve indicating that the maximum frequency could be obtained with no error in measurements is located in the range of frequencies between (4650-5500 Hz).

Figure (6) shows the obtained FRF curves for all models using Ansys software, figure (7) presents a sample of the mode shapes at all frequencies for all models and the value of deformation for one model, all data for all models was sorted in the table (2-4).

Table 2 - Comparison Between EMA and FEM Natural Frequencies (Hz) of the 6 Models

No.	20-20			25-25			20-25			30-30			25-30			20-30		
	FEM (Hz)	EMA (Hz)	Error %	FEM (Hz)	EMA (Hz)	Error %	FEM (Hz)	EMA (Hz)	Error %	FEM (Hz)	EMA (Hz)	Error %	FEM (Hz)	EMA (Hz)	Error %	FEM (Hz)	EMA (Hz)	Error %
1.	200	169	16	150	---	---	150	132	12	100	---	---	150	---	---	150	141	6
2.	400	419	5-	350	350	---	350	---	---	350	384	10-	300	---	---	350	434	24-
3.	550	425	23	500	547	9-	450	409	9	450	391	13	400	500	25-	550	669	22-
4.	650	659	1-	600	569	5	550	538	2	800	775	3	550	594	8-	700	794	13-
5.	850	800	6	750	847	13-	650	644	1	1000	978	2	650	825	27-	900	869	3
6.	1100	941	14	1050	1194	14-	750	809	8-	1350	1484	10-	1050	900	14	1100	1103	0
7.	1250	1319	6-	1300	1259	3	850	831	2	1500	1591	6-	1500	1578	5-	1200	---	---
8.	1350	1453	8-	1450	1453	0	1100	1175	7-	1900	1800	5	1900	1922	1-	1350	---	---
9.	1900	1972	4-	1600	1675	5-	1250	1213	3	2300	2291	0	2150	2184	2-	1800	1644	9
10.	2100	2181	4-	2150	2172	1-	1550	1475	5	2700	2759	2-	2350	---	---	2200	2113	4
11.	2450	2416	1	2500	2541	2-	2100	2031	3	2850	3016	6-	2500	2447	2	2500	2475	1
12.	2650	2631	1	2700	2728	1-	2300	---	---	3050	3063	0	2700	2719	1-	2700	2619	3
13.	2900	2913	0	3100	3034	2	2450	2494	2-	3300	3391	3-	2900	2897	0	2950	2931	1
14.	3400	3350	1	3550	3531	1	2700	2931	9-	3450	3406	1	3450	3428	1	3700	3638	2
15.	3750	3703	1	4300	3913	9	3100	3116	1-	3850	3872	1-	4300	4281	0	4200	4263	2-
16.	4000	3991	0	4950	4859	2	3400	3359	1	4150	4081	2	4650	4466	4	4550	---	---
17.	4200	4291	2-	5250	5053	4	3650	3684	1-	4450	4216	5	5000	5066	1-	4650	4872	5-
18.	4550	4503	1				3750	3703	1	4600	4531	2	5150	5303	3-			
19.	4750	4763	0				4000	4147	4-	4750	---	---						
20.							4150	4306	4-	4950	5253	6-						
21.							4300	4469	4-									
22.							4450	4556	2-									
23.							4650	4813	4-									
24.							4900	4944	1-									
25.							5400	5497	2-									
Average Error Ratio			4.98			4.68			3.46			4.32			4.98		4.68	4.68

Table 3 – FEM Deformation Values (mm) and Deformation Location of the 6 Models

No.	20-20		25-25		20-25		30-30		25-30		20-30	
	Deform Location	Max. Deform	Deform Location	Max. Deform	Deform Location	Max. Deform	Deform Location	Max. Deform	Deform Location	Max. Deform	Deform Location	Max. Deform
1.	Link	0.025457	Link	5.412200	Link	0.061916	Chuck	0.069340	Link	0.017443	Link	0.100930
2.	Platform	0.023780	Platform	9.609700	Platform	0.021897	Platform	0.013979	Platform	0.032198	Platform	0.039773
3.	Platform	0.017901	Link	4.300400	Link	0.007248	Platform	0.029920	Platform	0.006225	Platform	0.012773
4.	Link	0.008113	Link	4.115300	Tool Plate	0.017755	Link	0.002235	Tool	0.021965	Link	0.007091
5.	Platform	0.006642	Joint	2.494000	Link	0.007908	Tool	0.001650	Link	0.014760	Link	0.005946
6.	Platform	0.007043	Platform	0.407470	Joint	0.004858	Joint	0.000888	Platform	0.001431	Platform	0.005043
7.	Link	0.003465	Platform	0.282880	Link	0.006986	Platform	0.001548	Joint	0.000974	Platform	0.009527
8.	Link	0.003325	Joint	0.255400	Tool	0.005402	Joint	0.000326	Joint	0.000734	Joint	0.003964
9.	Joint	0.000947	Joint	0.152570	Link	0.012644	Joint	0.000284	Joint	0.002852	Platform	0.000899
10.	Joint	0.001517	Joint	0.672650	Platform	0.001497	Platform	0.000235	Joint	0.004834	Joint	0.000826
11.	Joint	0.000490	Joint	0.912620	Joint	0.001171	Platform	0.000248	Platform	0.000807	Joint	0.001650
12.	Joint	0.001300	Joint	0.484650	Joint	0.001521	Platform	0.000151	Platform	0.000264	Joint	0.000353
13.	Joint	0.000394	Joint	0.073029	Joint	0.000703	Joint	0.000108	Platform	0.000161	Joint	0.000270
14.	Joint	0.000372	Platform	0.068967	Joint	0.000314	Link	0.000283	Platform	0.000312	Link	0.000724
15.	Joint	0.000639	Platform	0.053603	Joint	0.000247	Joint	0.000076	Platform	0.000288	Link	0.000965
16.	Link	0.001698	Link	0.034014	Joint	0.000280	Joint	0.000187	Link	0.000351	Link	0.002563
17.	Link	0.003229	Joint	0.036449	Link	0.000556	Link	0.000663	Joint	0.000234	Link	0.000435
18.	Link	0.000218			Link	0.001037	Link	0.000324				
19.	Joint	0.000238			Link	0.000786	Link	0.000388				
20.					Link	0.005289	Joint	0.000282				
21.					Link	0.000678						
22.					Link	0.000372						
23.					Link	0.000700						
24.					Link	0.000245						
25.					Joint	0.000258						
Average Deformation		0.003682		1.727406		0.002705		0.006156		0.006248		0.011396

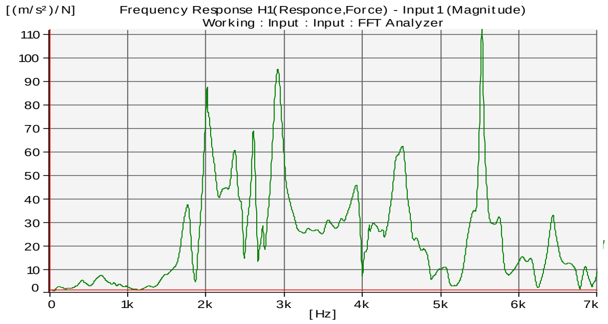

Figure (4-1) FRF of (20-20) System at Measuring Location (1)

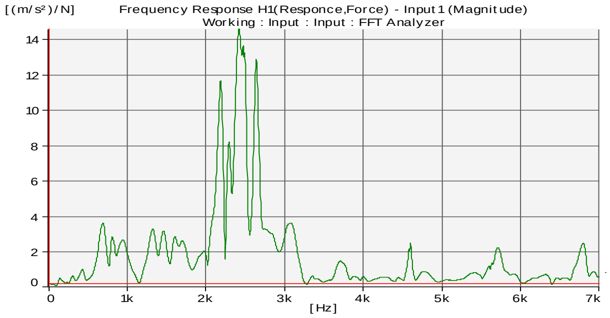

Figure (4-3) FRF of (20-20) System at Measuring Location (2)

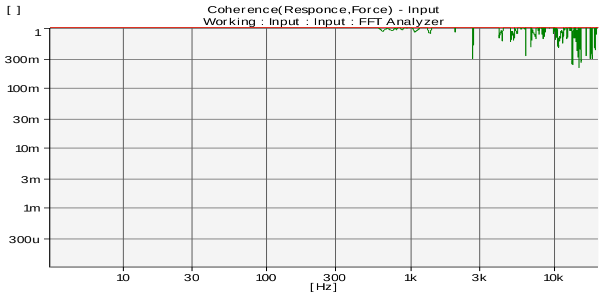

Figure (4-2) Coherence of (20-20) System at Measuring Location (1)

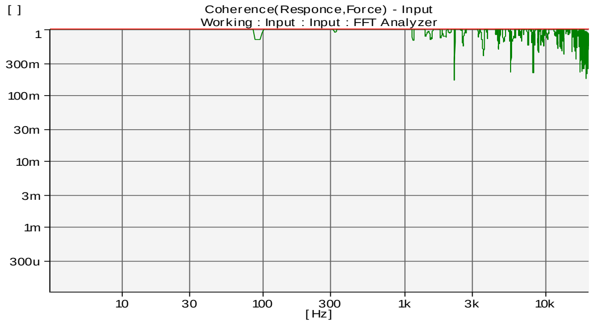

Figure (4-4) Coherence of (20-20) System at Measuring Location (2)

Figure 4 - EMA Measurements of (20-20) System

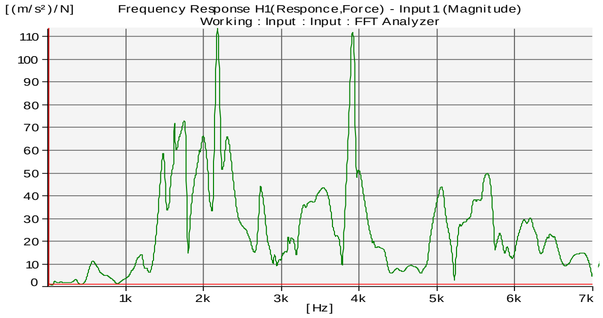

Figure (5-1) FRF of (25-25) System at Measuring Location (1)

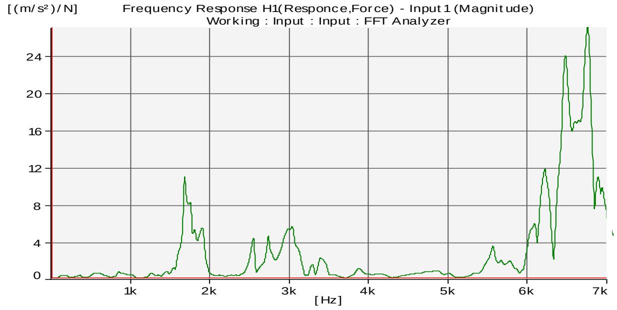

Figure (5-3) FRF of (25-25) System at Measuring Location (2)

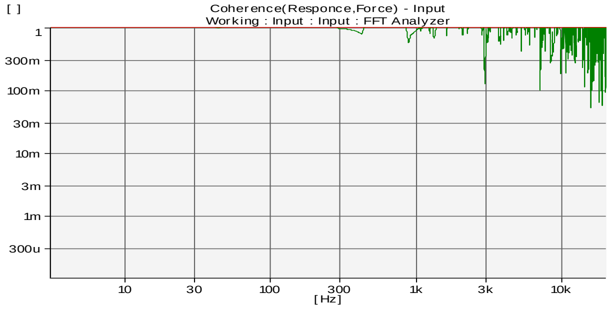

Figure (5-2) Coherence of (25-25) System at Measuring Location (1)

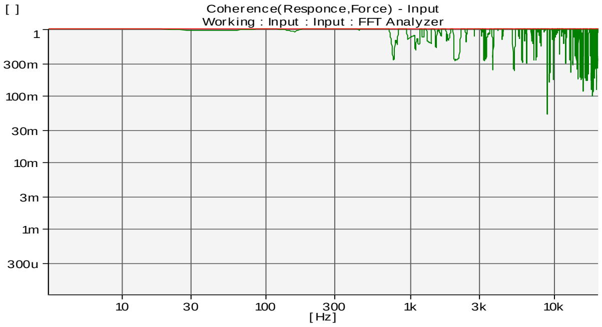

Figure (5-4) Coherence of (25-25) System at Measuring Location (2)

Figure 5 - EMA Measurements of (25-25) System

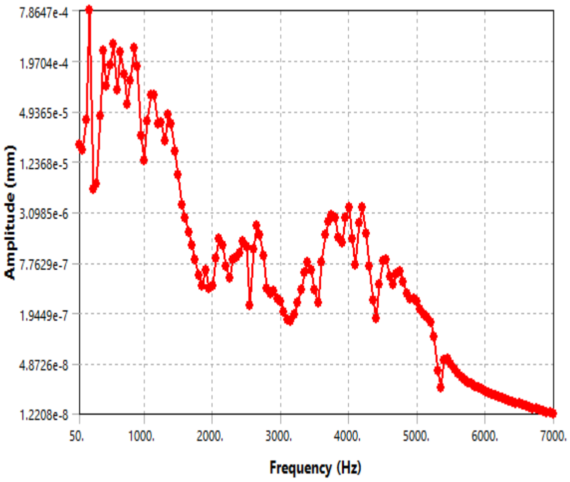

Figure (6-1) FRF of (20-20) Damped FEM System

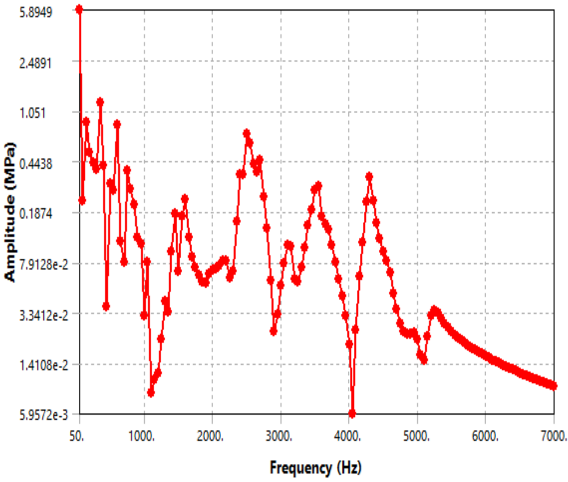

Figure (6-2) FRF of (25-25) Damped FEM System

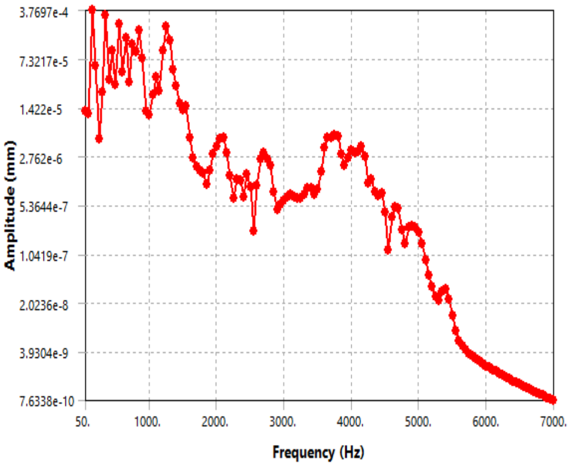

Figure (6-3) FRF of 25-20 Damped FEM System

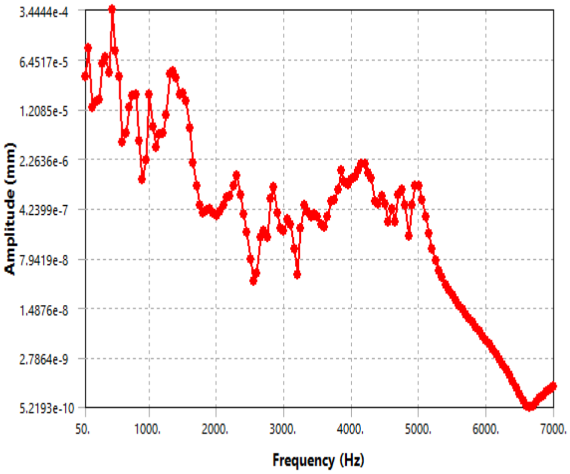

Figure (6-4) FRF of (30-30) Damped FEM System

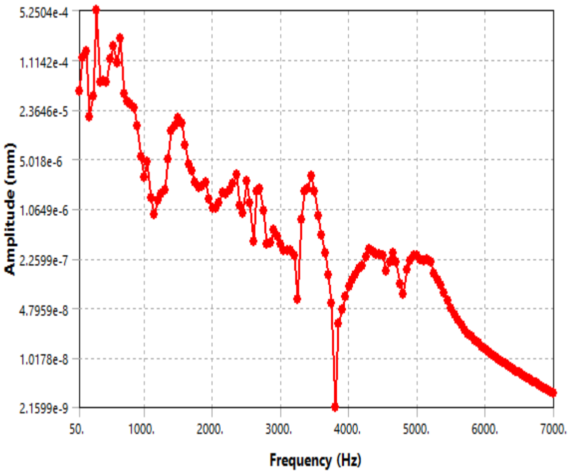

Figure (6-5) FRF of (30-25) Damped FEM System

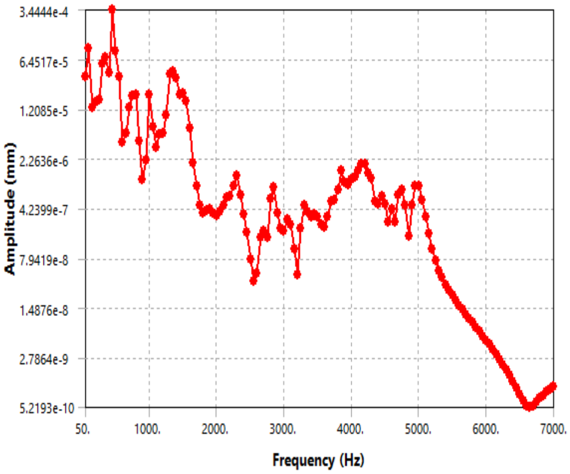

Figure (6-6) FRF of (30-20) Damped FEM System

Figure 6 - FRF of Damped FEM System

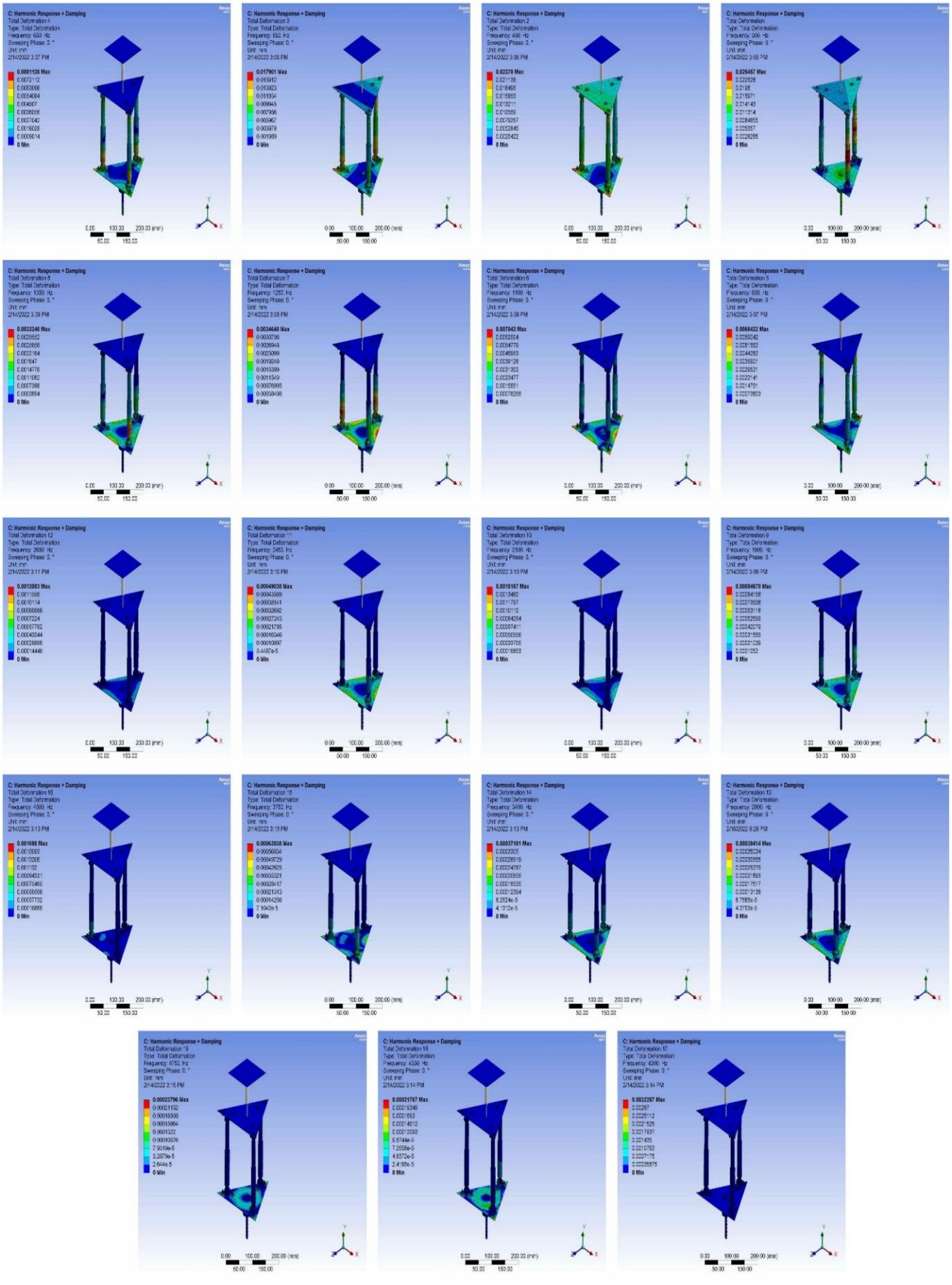

Figure 7 – Mode Shape of (20-20) Damped FEM System

Table 4 - Maximum Deformation Location for all Models.

Model Deformation Location	**20-20**	**25-25**	**20-25**	**30-30**	**25-30**	**20-30**	Σ Location
Link	7	4	13	5	3	7	39
Joint	8	8	8	7	5	5	41
Platform	4	5	3	6	8	5	31
Chuck	0	0	0	1	0	0	1
Tool	0	0	1	1	1	0	3
Σ Deformation	19	17	25	20	17	17	

6. Conclusion

In this paper, Experimental modal analysis and finite element analysis are used to analyse and evaluate the dynamic characteristics of parallel robot six models with three degrees of freedom, the results of the two methods were correlated, the natural frequencies, mode shapes, and deformation values were utilized to describe the design performance and evaluate the models, the following conclusion can be drawn:

- Finite element method is a robust tool to evaluate the natural frequencies of robot structure, the maximum value of the average ratio between experimental and Finite element method results didn't exceed 6.74%.
- Finite element method is a good tool to detect the missing modes that can't be detected in experimental work due to the high damping of components material and the lack of measuring software license.
- In a parallel robot, the maximum deformation occurs in links when the ratio between the platforms increases and appears in the joints when the platform's dimensions become equal.
- The results precision of finite element modeming increases by the accurate modelling of multibody system and considering damping of all components material.

6. References

[1] Chopra V., Smith A., and Angeles J., Jaber, A.A. and R. Bicker. A systematic strategy to find the natural frequencies of an industrial robot. in Proceedings of the International Conference on Advances in Mechanical and Robotics Engineering-AMRE. 2013.

[2] Elosegui, P., Measurement of the dynamic model of a PUMA 560 robot using experimental modal analysis. 1994.

[3] Berninger, T.F., et al., Evaluating the Mechanical Redesign of a Biped Walking Robot Using Experimental Modal Analysis, in Topics in Modal Analysis & Testing, Volume 8. 2022, Springer. p. 45-52.

[4] Wu, K. and B. Kuhlenkoetter, Experimental Analysis of the Dynamic Stiffness in Industrial Robots. Applied Sciences, 2020. 10(23).

[5] Mejri, S., et al. Experimental protocol for the dynamic modeling of machining robots. in Congrès français de mécanique. 2013. AFM, Maison de la Mécanique, 39/41 rue Louis Blanc, 92400 Courbevoie, France ….

[6] Glogowski, P., et al., Natural Frequency Analysis in the Workspace of a Six-Axis Industrial Robot Using Design of Experiments. Advanced Materials Research, 2016. 1140: p. 345-352.

[7] SHEN Guodong, N.J., TIAN Lili, EXPERIMENTAL MODAL MEASUREMENT AND DYNAMICS ANALYSIS OF A 6-DOFS SERIES ROBOT, in The 26th International Congress on Sound and Vibration - ICSV26. 2019: Montreal.

[8] JEEVAN, D.A.N.R., Modeling and Analysis of Robot ARM using ANSYS. International Journal of Scientific Engineering and Technology Research (IJSETR), 2015. 04(33): p. 6692-6697.

[9] Yu, Y., et al., Modal and harmonic response analysis of key components of robotic arm based on ANSYS. Vibroengineering PROCEDIA, 2017. 12: p. 109-114.

[10] Jain, R., M.N. Zafar, and J. Mohanta. Modeling and analysis of articulated robotic arm for material handling applications. in IOP Conference Series: Materials Science and Engineering. 2019. IOP Publishing.

[11] Nguyen-Van, S., et al., A novel modified analytical method and finite element method for vibration analysis of cable-driven parallel robots. Journal of Mechanical Science and Technology, 2020. 34(9): p. 3575-3586.

[12] Do, H. and K.-S. Park. Vibration Analysis of Cable-Driven Parallel Robot for Asymmetric Tension and Position. in Information Storage and Processing Systems. 2016. American Society of Mechanical Engineers.

[13] Tempel, P., et al., Application of the rigid finite element method to the simulation of cable-driven parallel robots, in Computational Kinematics. 2018, Springer. p. 198-205.

[14] Park, S.H., et al. ANSYS®-Based Gear Stress Analysis of a Lightweight 3-DOF Wrist Mechanism for a Parallel Robot with Expanded Workspace. in 2020 IEEE

International Conference on Mechatronics and Automation (ICMA). 2020. IEEE.

[15] Cooperative Modeling and Dynamics Simulation of Flexible Multi-Body System for Parallel Robot, in Proceedings of the 8th International Conference on Informatics in Control, Automation and Robotics. 2011. p. 321-324.

[16] Zhu, C.X., et al., Dynamics Simulation Analysis of Flexible Multibody of Parallel Robot. Applied Mechanics and Materials, 2007. 10-12: p. 647-651.

[17] Mohammed Ibrahim. Dawood, V.K., Design and Analysis of 3 - Axis Delta Type Parallel Robot. High Technology Letters, 2020. 26(10).

[18] Wang, X.-s., et al., Finite Element Modal Analysis of an Eight-axis Industrial Robot Painting System Applied to Boarding Bridge Painting, in Current Trends in Computer Science and Mechanical Automation Vol. 2. 2017, Sciendo Migration. p. 408-417.

[19] Rueda, J. and L. Ángel. Structural analysis of a delta-type parallel industrial robot using flexible dynamic of ANSYS 11.0. in 2009 35th Annual Conference of IEEE Industrial Electronics. 2009. IEEE.

[20] Sooraj Sukumar, K., et al., Kinematics, structural analysis and control of 3-ppss parallel manipulator. Int J Innov Technol Explor Eng, 2019. 8: p. 911-916.

[21] Luo, H.T., et al., Rigid-Flexible Coupling Dynamics Simulation of 3-RPS Parallel Robot Based on ADAMS and ANSYS. Applied Mechanics and Materials, 2013. 290: p. 91-96.

[22] Zhang, Q. and X. Zhang, Dynamic analysis of planar 3-RRR flexible parallel robots under uniform temperature change. Journal of Vibration and Control, 2013. 21(1): p. 81-104.

[23] Luo, H., et al., Rigid-flexible coupled dynamics analysis of 3-revolute-prismatic-spherical parallel robot based on multi-software platform. Advances in Mechanical Engineering, 2019. 11(3).

[24] Kumar, K.K., et al., Simulation and analysis of parallel manipulator for manoeuvring laparoscopic camera-CAD based approach. International Journal of Engineering and Technology, 2015. 7(1): p. 294-302.

[25] Liu, Z., et al. Comparison of finite element and experimental modal analysis of multi-joint flexible robotic arm. in 2017 International Conference on Mechanical, System and Control Engineering (ICMSC). 2017. IEEE.

[26] Li, H. and Z. Yang. Dynamic analysis of a parallel pick-and-place robot with flexible links. in Engineering Systems Design and Analysis. 2008.

[27] Toledo Fuentes, A., M. Kipfmueller, and M.Á. José Prieto. 6 DOF articulated-arm robot and mobile platform: Dynamic modelling as Multibody System and its validation via Experimental Modal Analysis. in IOP Conference Series: Materials Science and Engineering, 257. 2017.

[28] Huynh, H.N., et al., Modelling the dynamics of industrial robots for milling operations. Robotics and Computer-Integrated Manufacturing, 2020. 61.

[29] Tarabini, M., et al., Theoretical and Experimental Modal Analysis of a 6 PUS PKM, in Proceedings of the 16th International Conference on Informatics in Control, Automation and Robotics. 2019. p. 276-283.

[30] Palmieri, G., et al., Configuration-dependent modal analysis of a Cartesian parallel kinematics manipulator: numerical modeling and experimental validation. Meccanica, 2013. 49(4): p. 961-972.

[31] Zeng, Q., et al., Dynamic Characteristics Analysis of Two-Beam Laser Welding Robot for Fulselage Panels. 2021.

[32] Mahboubkhah, M., et al., Modal analysis of the vertical moving table of 4-DOF parallel machine tool by FEM and experimental test. Journal of Vibroengineering, 2017. 19(7): p. 5301-5309.