UDC 621
DOI: 10.15587/1729-4061.2022.252930

MODELING OF THERMAL DISTRIBUTIONS BY ANALYZING THE HEAT TOLERANCE OF A ROBOTIC GRIPPER PIVOT EXPOSED TO HEATED ELECTRONICS

Hasan Shakir Majdi
Professor Doctor, Dean
Department of Chemical Engineering and Petroleum Industries
Al-Mustaqbal University College
Hillah, Babil, Iraq, 51001

Atheer Raheem Abdullah
Assistant Lecturer
Department of Refrigeration and Air Conditioning Engineering
Al-Rafidain University College
Baghdad, Iraq, 10066

Auday Shaker Hadi
Doctor Lecturer
Department of Mechanical Engineering*

Laith Jaafer Habeeb
Corresponding author
Assistant Professor Doctor
Training and Workshop Center*
E-mail: 20021@uotechnology.edu.iq
*University of Technology – Iraq
Baghdad, Iraq, 10066

There are several applications in the aerospace, automotive and energy industries, for example, that often require high fidelity modeling or problems involving structural mechanics, heat transfer, or electromagnetic. Finite element analysis (FEA) is a popular method for solving the underlying partial differential equations (PDE) for these problems. 3D finite element analysis or 3D-FEA accurately captures the physics of these problems. The relevance of this study is to show how to set up finite element analysis (FEA) simulations and leverage the model of the environment to solve problems typically encountered by engineers and scientists in a variety of fields such as aerospace, automotive and energy. This study analyzes the behavior of mechanical components under different physical effects and shows a thermal analysis of a commercial KUKA YouBot robotic arm component by finding temperature distributions, figures, code, and test results for multiple materials. The developed model allows understanding and assessing the responsive component under loading, vibration or heat and determining deformation stresses among many things to select the best material and even prevent failure or undesired resonance as an example. These systems are typically modeled using partial differential equations or PDEs that capture the underlying physics of the problem and FEA is just one of the most common methodologies to solve this type of equation. The linear regression model can be a good predictive model that represents the relationship between thermal conductivity and max temperature to avoid undesired performance of the robotic arm

***Keywords:** finite element analysis (FEA), heat transfer, partial differential equations (PDE), robotic gripper pivot*

Received date 26.10.2021
Accepted date 13.02.2022
Published date 24.02.2022

***How to Cite:** Majdi, H. S., Abdullah, A. R., Hadi, A. S., Habeeb, L. J. (2022). Modeling of thermal distributions by analyzing the heat tolerance of a robotic gripper pivot exposed to heated electronics. Eastern-European Journal of Enterprise Technologies, 1 (1 (115)), 24–28. doi: https://doi.org/10.15587/1729-4061.2022.252930*

1. Introduction

There are a lot of different analytical methods that engineers can use to solve structural mechanics problems, whether it is to calculate the deflection of a beam or the stresses in a flat plate. The finite element method is a powerful numerical technique that uses computational power to calculate approximate solutions to these types of problems [1]. It is widely used in all major engineering industries. It could be used to check that satellite components will survive the launch conditions, for example. Finite element analysis software can be used to analyze a wide range of solid mechanics problems, including static, dynamic, buckling, and modal analyses. But it can also be used for fluid flow, heat transfer, and electromagnetic problems. For the study of the finite element method, we'll focus on how it applies to heat transfer analysis. Heat is the transfer of energy from objects of different temperatures, as objects warm up or cool down, their kinetic energy changes. Kinetic energy is the energy of motion. Temperature is a measure of the average kinetic energy of the particles in an object, as temperature increases, the energy of motion increases.

The paper [2] presented a comprehensive study in heat transfer FEA for mechanical, aeronautical, chemical, and nuclear engineering problems. There are several applications in the aerospace, automotive and energy industries, for example, that often require high fidelity modeling or problems involving structural mechanics, heat transfer, or electromagnetic. Finite element analysis (FEA) is a popular method for solving the underlying partial differential equations (PDE) for these problems. 3D finite element analysis or 3D-FEA accurately captures the physics of these problems as opposed to 2D-FEA or 1D-FEA [3].

Therefore, studies that are devoted to the FEA of heat tolerance for robotics are essential to analyze the efficiency and robustness even for a complicated design and provide information and leverage the model of the environment to solve problems typically encountered by engineers and scientists in a variety of fields such as aerospace, automotive and energy.

2. Literature review and problem statement

The paper [4] investigated the issue of thermal-mechanical behavior of selective laser melting (SLM) production, where the SLM process was modeled using stress and strain and heat transfer models. The thermal stress and 3D temperature distribution were simulated using a finite element analysis (FEA) coupled with the finite difference (FD) technique. Although the advantage of this approach is that the thermal-mechanical analysis can simulate the uneven temperature distribution throughout the part caused by breaking and warping problems, there were unresolved issues related to studying different materials and discussing the heat due to electronics components based on the simulated trials and thermal-mechanical analysis. The reason for this may be objective difficulties associated with robotic manipulators' complexity. On the other hand, the paper [5] used 1-dimensional and 2-dimensional models to simulate FEA for monolithic under radiant heating. Although this research discussed the FEA and thermal effects with specific emphasis in traditional uncoupled thermomechanical assessments, only two material thermal conductivities have been investigated based on experimental results accessible in the literature. The temperature distribution and the effect of heating conditions of also two materials, Types I and II structures of the power-sensitive strip, were investigated in [6]. A power probe heated at 50 mW was analyzed and modeled in thermal dynamics using the ANSYS program. This paper thoroughly discussed the thermal characteristics of the power density of the components by using temperature distribution, cooling, and heating rates theoretically and numerically, but the simulation time was too long. The research [7] discussed the heat transfer in FEA for directed energy deposition procedure. This study only compared the acquired results with the commercial software (SYSWELD). An option to overcome the relevant difficulties of the heat transfer in gas expansion processes can be still not fully understood, according to the study [8], since the expansion processes are complex and challenging due to the unsteady nature of the turbulent flow process.

The paper [9] utilized the Deep Learning technique to accurately predict heat distortion and tolerance restrictions by taking into account the local heat distribution for pointwise distortions. Although this approach gives accurate prediction and analyzes large data of numerous sensors, it doesn't provide a model allowing understanding and assessing the responsive component under heat. This issue has been discussed in [10] though studying the effect of surface temperatures on only one track geometry configuration through a process. The study quantifies the cause of surface temperatures on geometry to understand the effect of plane temperature on layer geometries and give a clear insight into heat geometry variations.

However, all these suggest that it is advisable to conduct a study to argue developing a model that allows understanding and assessing the responsive component under heat and determining deformation stresses among many things to select the best material and even prevent failure or undesired resonance as an example. This is done by analyzing the behavior of mechanical components under different physical effects and shows a thermal analysis.

3. The aim and objectives of the study

The study aims to perform a single-domain heat conduction analysis for a robotic gripper pivot exposed to heated electronics using MATLAB and partial differential equations modeling (PDEs). The practical expectations of these results can serve to find the associated acceptable heat-transfer coefficient for the max temperature of the alloy at which the robotic arm must shut off and avoid undesired performance of the robotic arm.

To achieve this aim, the following objectives are accomplished:

– to verify the visualized PDE-mesh model dimensions by comparing the estimated part sizes of the geometry with the manufacturer data of the robot gripper pivot component;

– to visualize the temperature distribution of the thermal model and test a magnesium alloy as a gripper pivot material;

– to model a relationship of thermal conductivity and the maximum temperature to avoid undesired performance of the robotic arm.

4. Materials and methods

The research hypothesis is to obtain an acceptable predictive model to provide a relationship between thermal conductivity and max temperature to avoid undesired performance of the robotic arm.

To determine the heat tolerance of a robot arm component, we analyze the behavior of mechanical components under different physical effects and show a thermal analysis of a robot arm component by finding temperature distributions, figures, and test results for multiple materials. It is possible to extend the analysis by leveraging the programming environment.

This model allows us to examine the responsive component under loading, vibration, or heat, and determine deformation stresses, among other things, so you can choose the appropriate material and even avoid failure or unwanted resonance. These systems are typically modeled using partial differential equations (PDEs) that capture the underlying physics of the problem and FEA is just one of the most common methodologies to solve this type of equation. Fig. 1 shows these steps with pictures.

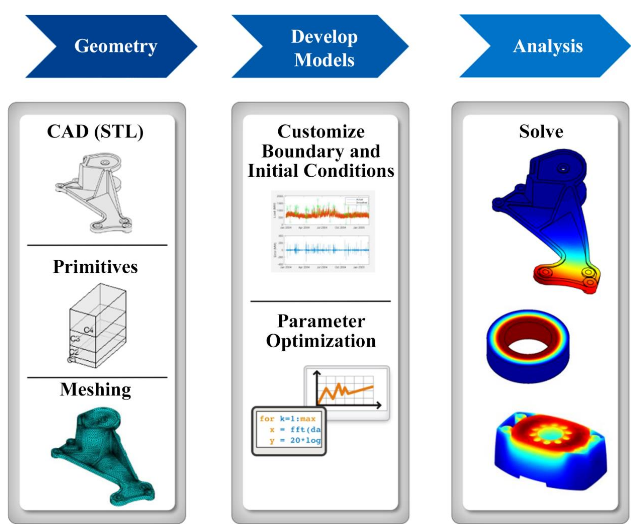
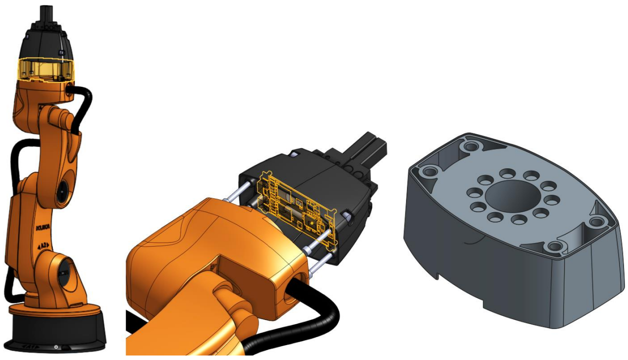


Fig. 1. Demonstration of the adopted methodology steps

First, we create or load geometry and generate a mesh. Second, we determine the type of model and moderate and initial conditions of the system. Third, we find the solution of the FEA and visualize and post-process the results. The FEA inspects the heat tolerance of a robotic gripper pivot exposed to heated electronics of a commercial KUKA YouBot robotic arm [11], which is shown in Fig. 2.



a *b* *c*

Fig. 2. Robotic gripper pivot exposed to heated electronics of the commercial KUKA YouBot robotic: *a* — full robot; *b* — robotic gripper pivot; *c* — pivot exposed to heated electronics

The analyzed part is a gripper pivot, which is part of a robotic arm. This pivot is exposed to heat through conduction of heat generated by a circuit board placed on top of it.

4. 1. Meshing the geometry of the material part

We define the type of model by solving the thermal steady-state problem.

We're going to import the geometry using an STL file, which can be generated through CAD packages.

Then, we use the PDE to visualize the geometry.

The next step is to generate a mesh based on the dimensions to be able to choose or decide on the appropriate element size for the tetrahedrons.

The bottom face of the robotic arm is kept at a constant temperature, assuming that the remainder of the arm works as a heat sink.

To depict the heat of the circuit board, heat flux is arbitrarily applied to the top face and inside the hole.

4. 2. Visualizing the temperature distribution of the thermal model

This stage includes setting up the boundary conditions, post-processing.

In the post-processing stage, the max temperature is calculated in Kelvin and converted to a more common unit such as Celsius or Fahrenheit.

The temperature distribution of the thermal model is visualized using the 3D-PDE plot of the MATLAB-based FEA tools with color map data argument, which also can measure the time of how long it takes to solve a single heat conversion.

4. 3. Best material selection and the relationship between thermal conductivity and max temperature

Here we extend this analysis to perform a design of experiments and parameter sweep to determine both the best material and to find a relationship between thermal conductivity and max temperature using additional materials. We use five additional thermal conductivities representing five additional materials and we are going to estimate how long it takes to run the simulation.

We are going to use a construct called (*parfor*), which stands a parallel for loop. This function allows using what is called MATLAB workers, which are essentially headless MATLAB sessions that are running in the background and are going to be carrying out all these computations simultaneously. Since each of these thermal problems is independent of each other, there's no reason to do one after the other. The results are stored and the max temperature for each of those thermal conductivities represents different materials.

5. Results of the developed modeling of thermal distributions

5. 1. Results of meshing the geometry of the material part

A MATLAB-based PDE geometry model was used to visualize the geometry as shown in Fig. 3.

The generated mesh representation is visualized using the PDE-mesh function as shown in Fig. 4.

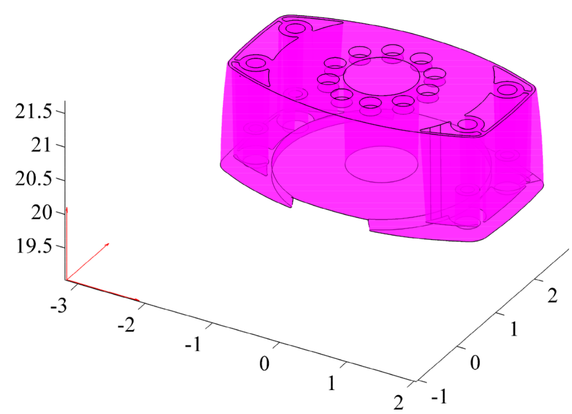


Fig. 3. MATLAB-based PDE geometry model of the gripper pivot

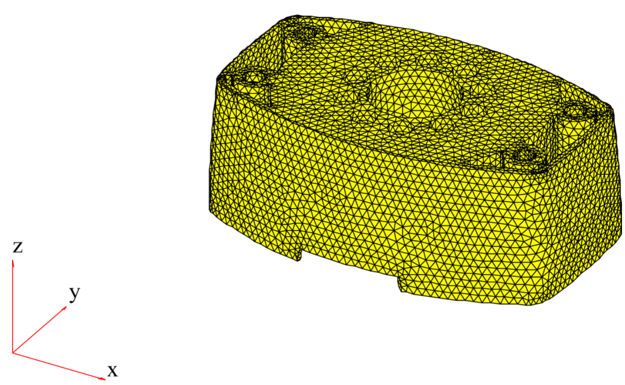


Fig. 4. Mesh visualization of the gripper pivot

In our application, the dimensions that were supplied by the manufacturer of this gripper pivot are about 32 millimeters [11]. Therefore, we've generated a mesh using a maximum element size for the mesh of 0.09 inches accordingly.

5. 2. Results of visualizing the temperature distribution of the thermal model

The boundary conditions assume that all the gripper pivot sizes are going to be exposed to the air temperature, so having natural convection of 25 (W/meter^{2}Kelvin), which also needed to be converted into W/inch^{2}Kelv to represent an ambient air temperature of 25 °C (288.1 K). The bottom part of the component is going to be attached to the robotic arm. Therefore, we are going to assume that is being held to a constant temperature while the inner circle and the top phase, which is closest to the circuit board, are going to have some heat flux.

Initially, we assume that the material is a magnesium alloy with a thermal conductivity of 52 (W/meter Kelvin), which is equivalent to 1.32 (W/inch Kelvin) [12, 13].

The thermal model has been solved in MATLAB-based FEA tools. We also measured the time of how long it takes to solve a single material, which takes about 31.1 seconds. The max temperature in the post-processing stage was calculated in Kelvin, Celsius, and Fahrenheit as 312.8 K=39.7 °C=103.5 °F, respectively.

To visualize the temperature distribution of the thermal model, a 3D-PDE plot with color map data argument was used as shown in Fig. 5.

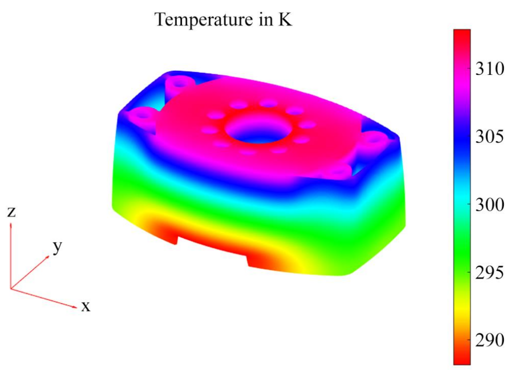


Fig. 5. Temperature distribution

Fig. 5 allows seeing how the temperature distribution on the bulk of the power is being transmitted. It is also found that the largest temperature is on the part that is closest to the circuit board and then as the heat moves along the power gets dissipated.

5. 3. Results of getting a relationship between thermal conductivity and max temperature to select the best material

Fig. 6 visualizes the relationships of thermal conductivities for all 5 types of materials with their maximum temperature.

It is found that there is a linear relationship that leads to the use of a linear regression model to find or create a predictive model that represents the relationship between thermal conductivity and maximum temperature. We've plotted this linear model to see how well this linear model adjusts to the acquired data as shown in Fig. 7.

The results obtained show that the max temperature is approximately equal to 330.41–0.342*thermal conductivity. This value is about 0.989 percent accurate based on the R-squared value, the results are shown in detail in Table 1.

Overall, we have a good fit for these five materials that we've considered. Now we're told that for this robotic arm, if it reaches the operating temperature of 311 Kelvin, the arm is going to shut off. Therefore, perhaps we want to design or select a material for this pivot so that the heat transfer doesn't allow for that kind of temperature to be reached.

In order to estimate the thermal conductivity needed to reach or to not reach 311 Kelvin temperature, we've applied the developed linear regression model. We see that thermal conductivity of less than 59.1 would keep the robotic arm operating under normal conditions, while any value that is larger than that is going to result in the robotic arm shutting out.

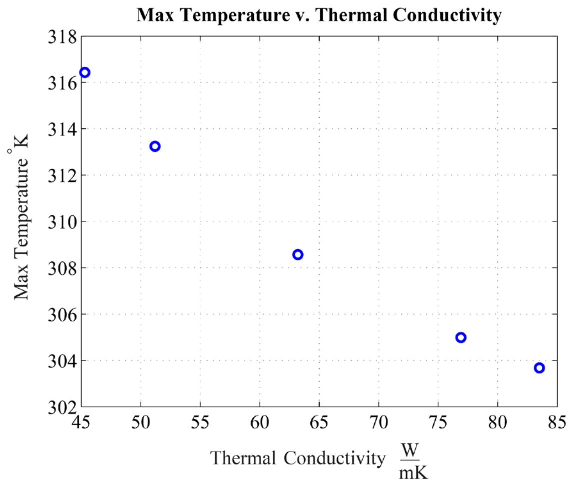


Fig. 6. Thermal conductivity represents the 5 materials under consideration vs. the max temperature of these materials

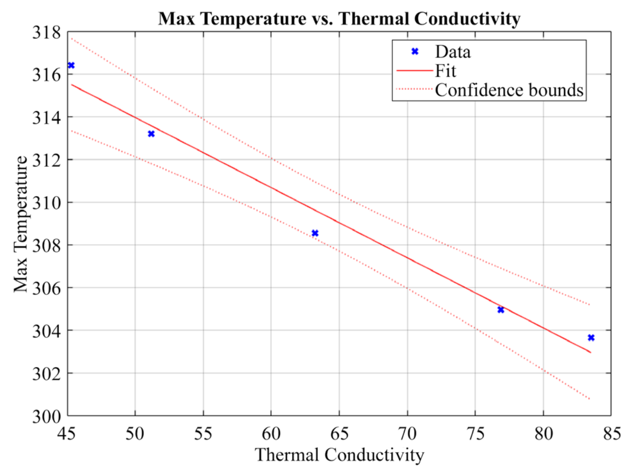


Fig. 7. Applying linear regression model on the obtained data of Fig. 7

Table 1

The estimated coefficients of the linear regression model

Model: maximum temperature ≈ 1+Thermal Conductivity				
Intercept	Estimate	SE	t-State	P-value
	330.41	1.88	175.75	4.0622e-07
Thermal conductivity	–0.328	0.0286	–11.475	0.00142
Number of observations: 5, Error degrees of freedom: 3				
Root Mean Squared Error:	0.933			
R-squared:	0.978			
Adjusted R-Squared:	0.97			
F-statistic vs. constant model:	132			
p-value:	0.00142			

6. Discussion of the results of the developed modeling of thermal distributions

The visualized geometry and PDE-based mesh models of the robotic gripper pivot, exposed to heated electronics, are shown in Fig. 3, 4. These models are verified by comparing the model dimensions with the manufacturer sizes data of the concerned component. The simulation results show that the part is about 4.032, 2.732, and 1.4606 inches, which is close to what the manufacturer determined (4.0, 2.71, and 1.46). These values represent the dimensions in x, y, z, respectively.

Referring to Fig. 5, the temperature distribution of the thermal model allows seeing how the temperature distribution on the bulk of the power is being transmitted. It is found that the largest temperature is on the part that is closest to the circuit board and then as the heat moves along the power gets dissipated. The max temperature in the post-processing stage was calculated in Kelvin, Celsius, and Fahrenheit as 312.8 K=39.7 °C=103.5 °F, respectively.

According to Fig. 6, which visualizes the relationships of thermal conductivities for all the 5 types of materials with their maximum temperature, a linear regression representation can be a good predictive model to represent this relationship as depicted in Fig. 7. The maximum temperature is approximately equal to 330.41–0.342*thermal conductivity. This value is about 0.989 percent accurate based on the R-squared value as listed in detail in Table 1.

The advantages of the developed model allow understanding and assessing the responsive component under loading, vibration or heat and determining deformation stresses among many things to select the best material and even prevent failure or undesired resonance as an example.

Two issues can be improved in the future; the more accurate regression model other than the linear one such as a polynomial model, and the computation time to solve the thermal model that can be improved using parallel loops to speed up the simulation. Such tools can parallelize the iterations crossways, parallel computing workers.

7. Conclusions

1. The obtained results prove that the visualized mesh model of the robotic gripper pivot was verified when the model dimensions with the manufacturer sizes data were convergent for the concerned component using MATLAB-based partial differential equations modeling tools (PDEs).

2. The color map data argument was used to visualize the temperature distribution of the thermal model of the robotic gripper pivot part, which allows seeing how the temperature distribution on the bulk of the power is being transmitted. It is found that the largest temperature is on the part that is closest to the circuit board, as the heat moves along the power gets dissipated.

3. Five additional thermal conductivities representing 5 different gripper pivot materials have been used to find the associated acceptable heat-transfer coefficient for the max temperature of the alloy at which the robotic arm must shut off. The linear regression model can be a good predictive model that represents the relationship between thermal conductivity and max temperature to avoid undesired performance of the robotic arm.

Acknowledgments

The authors gratefully acknowledge the Department of Chemical Engineering and Petroleum Industries, Al-Mustaqbal University College-Babil-Iraq for the support.

References

1. Lin, A.-D., Poon, S. Z., Tu, H.-W., Chen, C.-Y., Hsu, C.-M. (2020). Study on Thermoelectric Conversion and Conjugate Heat Transfer for PCBA by Finite Element Analysis. Applied Sciences, 10 (1), 197. doi: https://doi.org/10.3390/app10010197
2. Lewis, R. W., Morgan, K., Thomas, H. R., Seetharamu, K. N. (1996). The finite element method in heat transfer analysis. John Wiley & Sons, 350. Available at: https://www.wiley.com/en-us/The+Finite+Element+Method+in+Heat+Transfer+Analysis+-p-9780471943624
3. Ameri, M., Mansourian, A., Heidary Khavas, M., Aliha, M. R. M., Ayatollahi, M. R. (2011). Cracked asphalt pavement under traffic loading – A 3D finite element analysis. Engineering Fracture Mechanics, 78 (8), 1817–1826. doi: https://doi.org/10.1016/j.engfracmech.2010.12.013
4. Zhang, H., Xu, W., Xu, Y., Lu, Z., Li, D. (2018). The thermal-mechanical behavior of WTaMoNb high-entropy alloy via selective laser melting (SLM): experiment and simulation. The International Journal of Advanced Manufacturing Technology, 96 (1-4), 461–474. doi: https://doi.org/10.1007/s00170-017-1331-9
5. Kozłowski, M., Bedon, C., Honfi, D. (2018). Numerical Analysis and 1D/2D Sensitivity Study for Monolithic and Laminated
6. Structural Glass Elements under Thermal Exposure. Materials, 11 (8), 1447. doi: https://doi.org/10.3390/ma11081447
7. Ma, S., Wang, Z., Luo, J., Feng, T. (2015). Modelling and simulation of temperature distribution in 0.1 THz power probe. Zhenkong Kexue yu Jishu Xuebao/Journal Vac. Sci. Technol., 35 (09), 1064–1068. doi: https://doi.org/10.13922/j.cnki.cjovst.2015.09.06
8. Chua, B.-L., Lee, H.-J., Ahn, D.-G., Wang, Y. (2019). A Study on Activation Algorithm of Finite Elements for Three-Dimensional Transient Heat Transfer Analysis of Directed Energy Deposition Process. International Journal of Precision Engineering and Manufacturing, 20 (5), 863–869. doi: https://doi.org/10.1007/s12541-019-00118-9
9. Muhammad, A., Haruna, S. I. (2020). Finite element analysis of the heat transfer in a piston. Journal of Modern Manufacturing Systems and Technology, 4 (1), 45–51. doi: https://doi.org/10.15282/jmmst.v4i1.3722
10. Francis, J., Bian, L. (2019). Deep Learning for Distortion Prediction in Laser-Based Additive Manufacturing using Big Data. Manufacturing Letters, 20, 10–14. doi: https://doi.org/10.1016/j.mfglet.2019.02.001
11. Heilemann, M., Jothi Prakash, V., Beulting, L., Emmelmann, C. (2021). Effect of heat accumulation on the single track formation during laser metal deposition and development of a framework for analyzing new process strategies. Journal of Laser Applications, 33 (1), 012003. doi: https://doi.org/10.2351/7.0000307
12. KUKA youBot. EXPO21XX. Available at: https://www.expo21xx.com/industrial-robots/20317_st3_educational-robots/default.htm
13. Li, S., Yang, X., Hou, J., Du, W. (2020). A review on thermal conductivity of magnesium and its alloys. Journal of Magnesium and Alloys, 8 (1), 78–90. doi: https://doi.org/10.1016/j.jma.2019.08.002
14. Zhou, J., Yang, Y., Magne, L., Wang, G. (2001). Determination of thermal conductivity of magnesium-alloys. Journal of Central South University of Technology, 8 (1), 60–63. doi: https://doi.org/10.1007/s11771-001-0027-2