Strength analysis of parallel robot components in PLM
Siemens NX 8.5 program

P Ociepka and K Herbus

Silesian University of Technology, The Faculty Of Mechanical Engineering, Institute Of Engineering Processes Automation And Integrated Manufacturing Systems, ul. Konarskiego 18a 44-100 Gliwice, Poland

E-mail: piotr.ociepka@polsl.pl

Abstract. This article presents a series of numerical analyses in order to identify the states of stress in elements, which arise during the operation of the mechanism. The object of the research was parallel robot, which is the basis for the prototype of a driving simulator. To conduct the dynamic analysis was used the Motion Simulation module and the RecurDyn solver. In this module were created the joints which occur in the mechanism of a parallel robot. Next dynamic analyzes were performed to determine the maximal forces that will applied to the analyzed elements. It was also analyzed the platform motion during the simulation a collision of a car with a wall. In the next step, basing on the results obtained in the dynamic analysis, were performed the strength analyzes in the Advanced Simulation module. For calculation the NX Nastran solver was used.

1. Introduction

The dynamically changing demand on a market forces manufacturers to significantly accelerate designing and production of new products. It results that the engineers, who implement these processes, must use powerful informatics tools that are able to significantly speed up these processes while maintaining profitability, the proper quality and durability of the product. Modern programs of the CAD/CAE class [1-10] have great potential for carrying out advanced stress analysis, fatigue analysis or kinematic and dynamic ones. They allow simulating and analyzing many phenomena which occur during the operation of the mechanism. This makes possible to carry out a number of tests regarding the behavior of the virtual object in the virtual environment. Due to these analyses, it is possible to quickly perform a preliminary study of the considered object, what in turn allows limiting the scope, time and costs of researches conducted on real objects.

In the presented article is described the series of strength tests carried out for elements of the parallel robot, which is the basis of a driving simulator, created for people with disabilities (figure 1) [11,12,13,14]. These analyzes were performed in the NX 8.5 program. The aim of the analysis was to determine the maximal stresses and displacements that occur in elements of the parallel robot during operation of the simulator. For this purpose, during realization the dynamic analysis it was conducted, in the virtual model of the platform, the simulation of collision of a car with a wall. The simulation of platform operation was carried out in the Motion Simulation module. In the next step, basing on the results of the dynamic analysis, were performed strength analyzes in the Advanced Simulation module.
2. Identification of the trajectory and motion parameters of the parallel robot

In order to determine the maximal loads, which occur in elements of the parallel robot, was carried out the identification of the trajectory and motion parameters of the platform during the simulation of a car colliding with a wall. It is the most dynamic event that is simulated in the simulator. Therefore, the load of platform elements is then the highest. To determine the trajectory of the platform motion was used the fast camera Mega speed HHC x4. Two films were made which record, in two planes, the behavior of the selected points of the platform in the simulator space.

Processing and analysis of recorded films allowed establishing the approximate motion trajectory of the platform. In order to determine the motion parameters of the platform, during collision (velocity and acceleration), the measurements were realized using the Xsens Mtw Kit6 sensors. The analysis of recorded results allowed identifying the velocity and acceleration of the platform frame during the recorded movement. In figure 2 is shown the process of identification the trajectory and motion parameters of the simulator during the collision simulation.

The data, obtained in this way, were the basis for the dynamic analysis considered to the whole tested parallel robot.

Figure 1. Photo of the car simulator.

Figure 2. Identification of the motion parameters and trajectory during collision simulation.
3. Dynamic analysis in the Motion Simulation module

In order to conduct the dynamic analysis it was created the solid model of the parallel robot in the environment of the NX 8.5 software. In the Modeling module were created particular elements of the platform, and in the Assembly module was created the assembly of the model. The dynamic analysis was carried out in the Motion Simulation module. For calculation, were used the RecurDyn solver. The first step in preparing the model for the dynamic analysis was identifying the main components (links) involved in the conducted simulation of platform movement. It should be noted that one component may consists of many structural elements of the platform. The condition is that the elements, grouped in a single component (left), did not change the relative positions to each other during the simulation of platform movement. The prepared model of the whole system of the platform consists of seven components (links):

- **Base** - which consists of a foundation and attached to it bases with eyes to which the platform is attached;
- **Cross** - the lower joint component, which allows for the proper connection of the parallel robot leg with the base;
- **Engine** - which contains casing of an engine with all elements of a frame that allow for correct fitting of the engine;
- **Screw** - which consists of a screw with a frame and an end of a ball-and-socket joint;
- **Frame** - which consists of a frame, to which is fixed a car body and 6 slots of ball-and-socket joints;
- **Dr_sht** - a drive shaft, which ensures the transfer of power, from the engine to the nut of the screw, by means of a transmissions with a toothed belt;
- **Nut** - nut that allows transferring a torque on the screw of a screw transmission.

![Figure 3. Components created to conduct the dynamic analysis.](image)

The next step of the model developing was determination of joints (figure 4) between the particular components of the model to reconstruct the operation of the real object with the virtual model. For the implementation of the motion simulations the following types of joints were created [11,12]:

- **Base_fix** - joint of the *fixed* type that removes six degrees of freedom of the **Base** component,
- **Ki_Base** - joint of the *revolute joint* type allowing only for rotation of the **Cross** component around the chosen axis,
- **Ki_Mi** - joint of the *revolute joint* type allowing only for rotation of the **Engine** component around the chosen axis,
- **Mi_Wni** - joint of the *revolute joint* type allowing only for rotation of the **Dr_Sht** component around the chosen axis,
- **Mi_Ni** - joint of the *revolute joint* type allowing only for rotation of the **Nut** component around the chosen axis,
• *Screw* - joint of the screw type creating a screw transmission between the components **Nut** and **Screw**.
• *Sfr* - joint of the spherical type determining the possible motion between the components **Screw** and **Frame**.
• *Gear* - joint of the gear type determining the method of motion transmission from the **Dr_sht** component on the **Nut** component,
• *Slider* - joint of the slider type determining linear motion between trucks mounted on the **Nut** component and guides mounted on the **Screw** component.

Figure 4. Joints needed for obtaining the proper operation of the model.

In the next step were determined the motion characteristic reflecting the movement trajectory and motion parameters identified during the measurements of the real object. The dynamic analysis of the platform movement was carried out in order to determine the maximal forces that are applied to the components of the platform (forces actions in particular joints).

Figure 5. Results of the dynamic analysis.

In order to analyze the obtained results they were exported to the Microsoft Excel program, where they were processed. First was selected the leg of the platform, in relation to which the loads, during
the simulation, were the highest. And then was selected this step of the dynamic analysis during which was observed the maximal loads of the analyzed components of the platform. In the figure 5 are shown the simulation results of the analyzed movement of the platform. The results, obtained in this way, were then the basis for conducting the strength analysis with the FEM method.

4. Strength analysis in the Advanced Simulation module
The last stage of the realized research cycle was carrying out the strength analysis with the FEM method. This analysis was conducted in the Advanced Simulation module using the NX Nastran solver. It was analyzed this leg of the parallel robot in which occurred the highest loads. The model discretization, into finite elements model (TETRA 10) was done. It was defined material properties, introduced contacts between mating elements and defined appropriate joints (geometric constraints) in the model. The next step was to download the results of the dynamic analysis, in the form of forces acting on elements of the platform leg, and activating them in the FEM model. The program automatically introduced these forces (values, direction and senses) in the place where constraints were defined during realized dynamic analysis. Using such prepared FEM model were performed static strength analyses. The obtained results, in the form of contours of stresses and displacements, are shown in figure 6. It was found that the most enforced element is the ball-and-socket joint connecting the platform leg with the frame, to which is attached the vehicle body. The stresses occurring there (about 140 MPa) are not high what indicates that it is possible to increase the value of generated accelerations during the collision simulation. Further analyses showed that increasing the acceleration value of three times do not cause the risk of the joint damage. However, the generation of such acceleration during the motion simulation could be uncomfortable and dangerous for a simulator user.

![Figure 6. Graphical representation of the results of realized strength analysis.](image)

5. Conclusions
The use of advanced programs of the CAD/CAE class, during the design-constructional process, significantly speeds up creating a new product. The investigations let to determine the behavior of selected elements of the platform during operation of the virtual mechanism. The obtained results allow determining the motion parameters that could be obtained at the
maximal effort of the used components. The functioning simulator of car driving for the disabled persons could be seen on the website https://www.youtube.com/watch?v=D9yFqFPpWi4.

Acknowledgements
The work is realized within the project funded by The National Centre for Research and Development, agreement No UOD-DEM-1-495/001.

References
[1] Ociepka P and Świdner J 2004 Object-oriented system for computer aiding of the machines conceptual design process Journal of Materials Processing Technology 157-158 pp 221-227
[2] Ociepka P, Herbuś K and Gwiazda A 2014 Application of the CBR method for adding the design process of module manipulators Advanced Materials Research 1036 pp 1011-1016
[3] Herbuś K, Ociepka P and Gwiazda A 2014 Application of functional features to the description of technical means conception Advanced Materials Research 1036 pp 1001-1004
[4] Ociepka P and Grabowik C 2008 Zastosowanie metody CBR do wspomagania procesu projektowo-konstrukcyjnego i przygotowania technologii, Mechanik 8-9 pp 728-733 (in Polish)
[5] Gwiazda A 2014 System of designing complex technical means using fuzzy analysis App. Mech. and Mat. 474 pp 147-152
[6] Gwiazda A, Sękala A, Monica Z and Banaś W 2014 Integrated approach to the designing process of complex technical systems Advanced Materials Research 1036 pp 1023-1027
[7] Gwiazda A 2014 Design of the roof support with strait-line mechanism Advanced Materials Research 1036 pp 553-558
[8] Płaczek M 2014 Computer Aided Analysis of Systems with MFC Actuators Advanced Materials Research 1036 pp 711-714
[9] Ociepka P and Świdner J 2007 Współczesne systemy doradcze do wspomagania projektowania maszyn Monografia (Gliwice: Wydawnictwo Politechniki Śląskiej) (in Polish)
[10] Banaś W, Herbuś K, Kost G, Nierychlok A, Ociepka P and Reclik D 2014 Simulation of the Stewart platform carried out using the Siemens NX and NI LabVIEW programs Advanced Materials Research 837 pp 537-542
[11] Ociepka P, Herbuś K and Gwiazda A 2014 Application of the method basing on engineering knowledge and experience for adding the hexapod design process Advanced Materials Research 1036 pp 1005-1010
[12] Gwiazda A, Herbuś K, Kost G and Ociepka P 2015 Designing mechatronics equipment based on the example of the Stewart platform Solid State Phenomena 220/221 pp 419-422
[13] Gwiazda A, Herbuś K, Kost G and Ociepka P 2015 Motion analysis of mechatronic equipment considering the example of the Stewart platform Solid State Phenomena 220/221 pp 479-484
[14] Banaś W, Gwiazda A, Herbuś K, Kost G, Ociepka P and Reclik D 2015 Analysis of the dynamic properties of the mechatronic integrator of control procedures of the vehicle driven by persons with disabilities Solid State Phenomena 220/221 pp 3-8