GDAŃSK UNIVERSITY OF TECHNOLOGY

Subject card

| | | | |
|---|---|---|---|
| Subject name and code | Computer Aided Engineering - CAE, PG_00024951 | | |
| Field of study | Medical and Mechanical Engineering, Medical and Mechanical Engineering | | |
| Date of commencement of studies | October 2020 | Academic year of realisation of subject | 2022/2023 |
| Education level | first-cycle studies | Subject group | Optional subject group<br>Subject group related to scientific research in the field of study |
| Mode of study | Full-time studies | Mode of delivery | at the university |
| Year of study | 3 | Language of instruction | Polish |
| Semester of study | 6 | ECTS credits | 4.0 |
| Learning profile | general academic profile | Assessment form | assessment |
| Conducting unit | Department of Machine Design and Vehicles -> Faculty of Mechanical Engineering and Ship Technology | | |
| Name and surname of lecturer (lecturers) | Subject supervisor | dr inż. Leszek Dąbrowski | |
| | Teachers | dr inż. Leszek Dąbrowski | |

**Lesson types and methods of instruction**

| Lesson type | Lecture | Tutorial | Laboratory | Project | Seminar | SUM |
|---|---|---|---|---|---|---|
| Number of study hours | 15.0 | 0.0 | 30.0 | 0.0 | 0.0 | 45 |

E-learning hours included: 0.0

**Learning activity and number of study hours**

| Learning activity | Participation in didactic classes included in study plan | Participation in consultation hours | Self-study | SUM |
|---|---|---|---|---|
| Number of study hours | 45 | 5.0 | 50.0 | 100 |

**Subject objectives**

The ability of structural analysis of machine parts in terms with Finite Elements Method (FEM). Getting to know the steps and elementar methods applied in professional calculation systems and classic mechanical problems.

**Learning outcomes**

| Course outcome | Subject outcome | Method of verification |
|---|---|---|
| K6_W09 | Student themself builds a geometrical model of construction in ANSYS computing system. He takes into account model parameters. They split the geometrical model of construction to finite elements: shell and solid. They define the support and the required nodal, line and surface loads. They implement selection procedures to define support and load. They analize the safety and rates the linear stiffness with usage of surface and volume elements. They use large deformation technique to estimate the load limit connected with loss of the stability of the structure. They locate and eliminate errors in program written in APDL language. They build flat and axially-symetric models themself. They apply the model of plasticity of the material. They build models of interactions of parts taking into account contact finite elements. | [SW3] Assessment of knowledge contained in written work and projects |
| K6_U05 | Student draws conclusions from the results of FEM calculations, evaluates the risk of various forms of machine part destruction based on FEM. | [SU1] Assessment of task fulfilment |
| K6_U03 | The student understands the problem presented in the drawing, compares his solution with fellow students solving a simmilar problem. | [SU1] Assessment of task fulfilment |

<table>
<tr><td>Subject contents</td><td>LECTURE<br>Methods of describing the shape in FEM programs. Methods for the automatic distribution of a geometric model construction for shell and solid elements. The possibilities and advantages of the parametric description of a geometric model, and load support in a FEM model. The course analyzes the state of stress and linear stiffness assessment, differences in the use of linear, surface and volume elements. Models of plasticity of material and its description in FEM program. The method of load limit disclosure. The method of taking into account the impact of strain on the stiffness (taking into account large deformations). Goals and possibilities of co-modeling many of parts of the machine. Discussion of the elements of contact and procedures for the detection range of the contact. Possibility to include friction in the model of numerous parts of the machine and methods of friction results observation in the calculations. Distributed load and mass forces modeling capabilities.<br><br>LABORATORY EXERCISES<br>Running sample programs in ANSYS APDL computing system, describing FEM models illustrating the thesis of the lecture. Self-building of six individual computing tasks related with: solid modeling, parametric modeling, modeling study of shell stability loss, modeling with the model of a plasticity, contact model of two parts, improving the design based on the results of the calculation results in a graphical environment.</td></tr>
<tr><td>Prerequisites and co-requisites</td><td></td></tr>
<tr><td>Assessment methods and criteria</td><td><table>
<tr><th>Subject passing criteria</th><th>Passing threshold</th><th>Percentage of the final grade</th></tr>
<tr><td>six projects</td><td>33.0%</td><td>100.0%</td></tr>
</table></td></tr>
<tr><td>Recommended reading</td><td><table>
<tr><td>Basic literature</td><td>Course Website: http://www.kkiem.mech.pg.gda.pl/oacm/kwpi/</td></tr>
<tr><td>Supplementary literature</td><td>-Zagrajek T., Krzesiński G., Marek P.: Metoda elementów skończonych w mechanice konstrukcji. Ćwiczenia z zastosowaniem systemu ANSYS. Oficyna Wydawnicza Politechniki Warszawskiej 2005</td></tr>
<tr><td>eResources addresses</td><td></td></tr>
</table></td></tr>
<tr><td>Example issues/ example questions/ tasks being completed</td><td>Check the strength of the structure shown in Figure.</td></tr>
<tr><td>Work placement</td><td>Not applicable</td></tr>
</table>