Module Title: 4B7 Computer Aided Engineering

Code: ME4B7

Level: Senior Sophister (Optional module)

Credits: 5

Lecturer(s): Assistant Professor Seamus O'Shaughnessy*
Professor David Taylor
Assistant Professor Tim Persoons

* Module Coordinator – Email: oshaugse@tcd.ie

Module Organisation
This module runs for the 12 weeks of semester one (except during study/assignment week) with assignment activity and submission dates in semester one. Two formal lecture slots are available per week but the programme is predominantly computer lab based. Total contact time is 50 hours.

<table>
<thead>
<tr>
<th>Semester</th>
<th>Start Week</th>
<th>End Week</th>
<th>Associated Practical Hours</th>
<th>Lectures (per week)</th>
<th>Lectures (total)</th>
<th>Tutorials (per week)</th>
<th>Tutorials (total)</th>
<th>Contact hours (total)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>12</td>
<td>17</td>
<td>2</td>
<td>22</td>
<td>1</td>
<td>11</td>
<td>50</td>
</tr>
</tbody>
</table>

Module Description, Aims and Contribution to the Programme
The module is centred on the application of a complex commercial finite element programme to address a number of design problems in engineering. These will include stress analysis, heat transfer, fluid mechanics, vibration, and contact problems.

Learning Outcomes
On completion of this module, the student will be able to:

1. Complete an analysis cycle from drawing to calculation of a component
2. Interface a finite element analysis with a CAD package
3. Perform various types of mechanical engineering analysis
4. Implement a design cycle
5. Operate a commercial finite element package
6. Understand and interpret results of finite element analysis and know how to verify and optimise the calculation procedures
Module Syllabus

- Geometry Input/CAD interface
- Stress Analysis
- Contact Analysis
- Non-Linear and iterative calculation procedures with time step control
- Vibration Analysis
- Heat Transfer (Static and Dynamic)
- Thermal stress problems

Teaching Strategies

This module is taught primarily through assignment with supporting lectures and tutorials. An initial assignment will be presented to enable problem formulation followed by a linear stress analysis. The function of this will be to establish working familiarity with the package. Three distinct design challenges will then be presented working in different areas of engineering.

Recommended Text(s)

- ANSYS Training materials ... available in .pdf

Assessment

This module is assessed by reporting of results as assigned using a combination of package generated reports, log files and oral presentations (if appropriate to a particular assignment). 60% will be awarded for the reporting element. In addition, individual testing will occur in a computer lab environment where it will be expected that simplified assignments are completed within a timed period. This will account for 40% of the final mark.

Experiment/Software

ANSYS-WORKBENCH 14.x