MTE 491/591: Theory and Applications of CFD in Materials Processing

Prerequisites

None

Course Description

This course will cover the Computational Fluid Dynamics (CFD) methodology using Ansys Fluent. Students will learn how to use Ansys Fluent in a computer classroom setting to solve fundamental heat and mass transfer problems in Materials Processing.

Outline of Main Topics

• Introduction to CFD Fluent
  • Ansys Designer (CAD Geometry)
  • Ansys Mesher (for CFD)
  • Heat Transfer modeling using Fluent
  • Natural convection
  • Turbulence modeling
  • Multiphase modeling in Fluent
  • UDF in Fluent

Text Material

1. Handouts (Available in Blackboard)


Evaluation and Grading

<table>
<thead>
<tr>
<th>GRADE DETERMINED AS FOLLOWS:</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>40% - average of 3 exams</td>
<td>A 90</td>
</tr>
<tr>
<td>60% - special project</td>
<td>B 80</td>
</tr>
<tr>
<td></td>
<td>C 65</td>
</tr>
<tr>
<td></td>
<td>D 50</td>
</tr>
</tbody>
</table>