

FDP-25	<b>CIRCUIT SIMULATION AND PCB DESIGN USING MULTISIM</b>	<b>10.06.2019 to 14.06.2019</b>
<p><b>OBJECTIVES:</b></p> <ul style="list-style-type: none"> <li>➤ Draw the electronic circuit using Multisim</li> <li>➤ Simulate Electronic Circuits using Multisim</li> <li>➤ Design PCB Layout</li> <li>➤ Interface Multisim with LabVIEW</li> </ul> <p><b>PARTICIPANTS:</b></p> <p>Teachers of Electrical, Electronics and Instrumentation who have working knowledge in Electronic circuits.</p> <p><b>INPUT:</b></p> <p>Symbols – Library of components - Schematics – Component Editor – Simulation – DC bias, DC and AC sweep analysis – Transient Analysis – Parametric Analysis - Simulation of Digital and Analog Circuits – PCB Design - Layout - Net List – Layers – Routing – Electrical Rule Check (ERC) – Single side and double side PCB layout – Interfacing Multisim with LabVIEW- Bill of Materials – Technical Documentation.</p> <p><b>PROCESS:</b></p> <ul style="list-style-type: none"> <li>➤ Lecture</li> <li>➤ Demonstration</li> <li>➤ Laboratory sessions</li> <li>➤ Industrial Visits</li> </ul> <p><b>OUTPUT:</b></p> <p>The Participants will be able to handle electronic circuits course and PCB design using Multisim software.</p> <p><b>RESOURCE PERSONS:</b></p> <ul style="list-style-type: none"> <li>➤ Dr. P. Sivasankar</li> <li>➤ Guest Faculty</li> </ul>		
<b>COORDINATOR</b>	<b>VENUE</b>	<b>LAST DATE FOR RECEIPT OF APPLICATIONS</b>
Dr. G.A. Rathy	NITTTR, Chennai	30 days prior to the start of the programme