

„Moderno tehnoloģiju fizikālo procesu datormodelēšana” (24. - 27. janvāris)

Block 2: Solving simple mechanical and fluid problems using ANSYS and CFX



Time	24.01.2007	25.01.2007	26.01.2007	27.01.2007
09:00 – 09:45		<b>Lecture:</b> Introduction to CFX - Capabilities - Boundary Conditions - Post Processing	<b>Lecture:</b> Introduction to Ansys - Capabilities - Boundary Conditions - Post Processing	<b>Demonstration / Exercise:</b> Data management - Using environments - Comparing designs - Generating reports
09:45 – 10:30		<b>Exercise:</b> Tesla valve - Setup flow simulation with CFX-Pre - Run CFX Solver - Analyses results with CFX-Post	<b>Exercise:</b> Circular Plate with Holes - Create geometry and mesh - Setup structural simulation - Solve and analyse results	
10:30 – 11:00		<b>Coffee break</b>	<b>Coffee break</b>	
11:00 – 11:45		<b>Exercise:</b> Tesla valve - Find characteristic curve (pressure drop vs. flow rate) for forward and backward flow	<b>Exercise:</b> T-Junction - Import CAD geometry - Clean-up CAD model	<b>Laboratory Work:</b> Mechanical, thermal and/or fluid flow problems
11:45 – 12:30		<b>Exercise:</b> T-Junction - Import CAD geometry - Extract of fluid domain - Add geometry		
12:30 – 13:00			<b>Lunch time</b>	<b>Lunch time</b>
13:00 – 14:00	<b>Introduction</b>			<b>Open Forum:</b> - Questions - Suggestions - Outlook - Individual projects
14:00 – 14:45	<b>Lecture:</b> Overview of Ansys Workbench - DesignModeler - Ansys - CFX	<b>Exercise:</b> T-Junction - Setup flow simulation - Run Solver - Postprocess results	<b>Exercise:</b> T-Junction - Add loads (thermal/mechanical) - Run solver - Postprocess results	
14:45 – 15:30	<b>Demonstration:</b> Solving structural, thermal and fluid flow problems in ANSYS Workbench			
15:30 – 16:00	<b>Coffee break</b>	<b>Coffee break</b>	<b>Coffee break</b>	
16:00 – 16:45	<b>Exercise:</b> Tesla valve - Create CAD model with DesignModeler	<b>Demonstration / Exercise:</b> Parametric analysis - Change dimensions of model - Update geometry - Re-run flow simulation	<b>Laboratory Work:</b> Mechanical, thermal and/or fluid flow problems	
16:45 – 17:30	- Generate unstructured meshes with CFX-Mesh			
17:30 – 17:45	<b>Break</b>	<b>Break</b>	<b>Break</b>	
17:45 – 18:30	<b>Laboratory Work:</b> Geometry generation and meshing	<b>Laboratory Work:</b> Fluid flow and heat transfer problems	<b>Laboratory Work:</b> Mechanical, thermal and/or fluid flow problems	