CATIA V5 Fundamentals Training Certificate

Lawrence Technological University

College of Engineering

CATIA is one of the world's leading high-end integrated CAD/CAM/CAE systems. CATIA V5 is a state of art solid modeling software used in automotive and aerospace industry by companies like: Chrysler, Toyota, BMW, Bombardier, Lockheed and Boeing. It is extensively used by many mould making, tool and die, and sheet metal companies.

The emphasis of this training is on the graphical communication of solutions to mechanical engineering design problems. This training should provide a solid background for solid modeling and assembly for digital product development. By the end of this CATIA Fundamental Training, participants should be able to apply the fundamentals of solid modeling including feature based parametric CAD by completing a three dimensional part; explain the engineering design process and the role of engineering graphics in digital design and analysis; create a 3D assembly model, including a bill of materials; demonstrate the management of CAD data files and integrated data management documentations; and understand the product lifecycle management (PLM).

Training Outlines:

Lecture	Topics
1	Introduction to CATIA, 2D Sketch and Dimensioning
2	Operations on Profiles, and Introduction to 3D Model
3	3D Solid Model using Protrusion, Revolution, Loft, Sweep, etc.
4	3D Complex Geometry
5	3D Assembly Model
6	3D Assembly, Engineering Document Management, and Product Lifecycle Management

Instructor Information:

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Lecture: Saturday, 8:00 am - 12:00 pm
Room: E30 (Engineering Building) or E152
Duration: Six weeks
Course Fee: $1,200 for LTU Students ($1,500 for outsiders)

Certificate:

After successful completion of the training, a certificate will be provided.

References:

CATIA V5 Workbook, Release 17, Richard Cozzens, SDC Publication, 2007, ISBN: 978-1-58503-399-7.
CATIA V5, National Institute for Aviation Research, Wichita State University, 2007.