



Siemens NX 10 Surface Design: A Step by Step Guide

By Jaecheol Koh



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This textbook explains how to create freeform surface and modify them to create freeform face of a solid body using Siemens NX 10. NX is a three dimensional CAD/CAM/CAE software developed by Siemens PLM Software Inc., Germany. This textbook is based on NX 10.0. Users of NX 9.0 can use this book with minor modifications. We provide files for exercises via our website. All files are in NX 6.0 so readers can open the files using NX 6.0 and later releases. It is assumed that readers of this textbook understand basic modeling process with NX. He/She has to be able to create sketch and fully constrain it, create the extruded and revolved features, apply boolean operation between solid bodies and understand how to use part navigator and selection toolbar. This textbook is suitable for anyone interested in creating mechanical surface and applying for solid body using Siemens NX. Topics covered in this textbook - Chapter 1: Basic components of Siemens NX 8.x, options and mouse operations. - Chapter 2: Introduction to surface modeling process of NX 10. - Chapter 3 and 4: Creating Ruled and Through Curves surface. - Chapter 5: Face analysis. - Chapter 6, 7 and 8: Creating Through Curve Mesh, Swept and Variational Sweep surface. - Chapter 9: Commands for creating curves. - Chapter 10: Other helpful commands for creating surface model. - Chapter 11: Modeling projects. - Chapter 12: Modeling Bumper Surface of Audi Q5

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