

Basic and Intermediate

NX5

Modeling, Drafting, and Assemblies

A Project Oriented Learning Manual

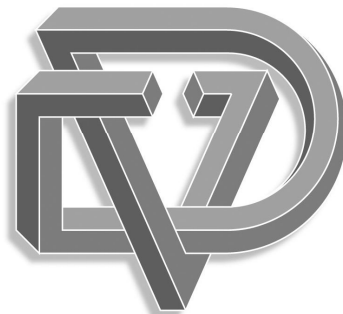
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**DESIGN
VISIONARIES**



**Superior Vision Yields
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Dedication

We dedicate this book to David F. Samuel. Many people have been touched by his creativity, wisdom, and love. We will miss him.

Acknowledgements

We would like to thank the following people for their tireless efforts. Without the contributions from each of you this book would be a mere shadow of what it has become:

Rick Uro, Ben Stevenson, Charlene Brum, Julie Gantner, Melissa Sweat, Craig Dorety, and Jim Defever

What readers have to say about our previous books:

Bryan McDonald, Product Design Manager at Apple Computers

“Bottom line: this is an excellent book. If anyone wants to learn how to use Unigraphics quickly, efficiently and practically, this is the way to do it.”

Fred Dyen, Director of St. Louis University’s Aviation Maintenance Institute (AMI)

“Practical Unigraphics NX2 Modeling for Engineers was extremely effective and much better than [other] textbooks. I would highly recommend this book to other professors and students alike.”

Dr. Pat Spicer, Professor at Western Illinois University

“The UG NX2 textbook is well organized. Its tutorial style of learning is easy for students to utilize. The practice exercises are essential. From my experience in teaching students to use UG software, I have found that this is the best textbook currently on the market for teaching UG NX2.”

Preface

Dear reader,

Thank you for purchasing our NX5 Basic and Intermediate book, the newly revised version of our previous CAD training text books. We have greatly expanded the content, detail, and exercises included in this edition. Design Visionaries is an engineering consulting firm that performs many design projects great and small, including industrial design, product design and engineering analysis. Our customers entrust us with the design of medical devices, aerospace components, heavy machinery, consumer products, etc. The methods outlined in this book go beyond an academic use of the software. They are tricks of the trade that come from thousands of hours of actual use of the software to design some of the most difficult products in the world. In addition, Design Visionaries offers world class on-site training which enables us to develop and evolve our training material so that they provide the maximum benefit. Please enjoy this text, and we invite you to log on to our website – www.designviz.com where you can find additional free materials, other advanced materials, products, and goodies.

Thank you,

Stephen Samuel

September 07, 2007

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1. An Overview of NX5

1.1 The Philosophy of this Book

This book has been written with a certain underlying philosophy that comes from years of design engineering which we would like to share with you. Engineers are pretty bright in general, so we've written this book to take advantage of that fact. Our book begins with basics and examples explained to every last detail. As the book progresses, more and more is left to the reader. We believe this enables faster learning because you won't have to sift through copious and superfluous instructions. We hope you enjoy this material that we've truly poured our hearts and souls into.

Using NX5 is like playing a piano. In the same way that chords are as important as individual notes, NX commands are far more powerful when used in concert with others. Our book makes an effort to show not only the details of the most important commands, but the powerful combinations that we have used to bring about excellent designs.

All projects and practice exercises are included on the CD-ROM. Use them for your reference or as an example of how each model/project should look upon completion.

1.2 Basic Concepts of NX5

UGS NX5™ (commonly referred to as “Unigraphics”, “UG”, or “NX5”) is a full-featured 3D Computer-Aided Design (CAD) software that is used by companies all over the world to design state-of-the-art products. This manual teaches you the basic modeling, assemblies, and drafting functionality of NX5. In NX5, each of these functions is organized into a discrete “Application”.

Think of NX as a suite of applications; each of which has some tools specific to it, and other tools which are shared with other applications. When you initially start NX5, you enter what is known as the “Gateway” Application. It allows you to examine the geometry and drawings that have been created, but most of the functionality is not available. The most common approach to CAD design is to first create a three-dimensional (3D) geometric model, and then create the supporting engineering documentation. During the course of a design project, you will most frequently use the Modeling and Drafting Applications (see below).

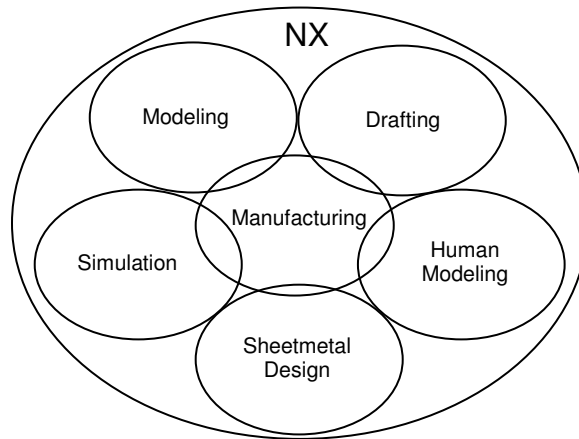


Figure 1

When using NX5, the first decision you must make is which of the various Application modules to invoke. The Modeling Application is used to define 2D and 3D wire frame geometry, as well as solid models. The Drafting Application is used to create the orthographic views, dimensions, notes, and other information required for engineering documentation purposes. NOTE: In NX5, the part model and the drawing can be combined into one CAD database file. Another alternative is to use the Master Modeling Philosophy where a separate CAD database file is created for each purpose. In this strategy, the non-master drawing looks back to the master part model.

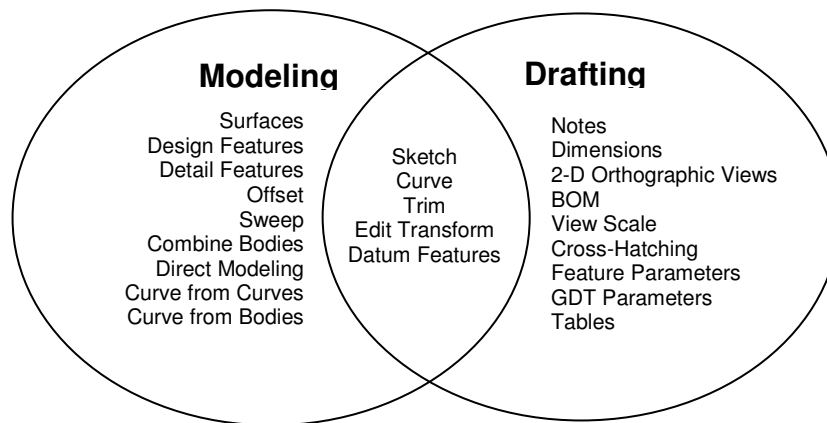


Figure 2

This tutorial concentrates on what is typically the most complicated task in NX5: 3D geometric modeling. The above figure diagrams some of the primary tools (or functions) for creating geometry within the Modeling and Drafting applications, and some of the tools they ‘share’ between them.

Before continuing the discussion of how to use the NX5 modeling tools, it is useful to distinguish between ‘Parametric’, ‘Non-Parametric’, and ‘Hybrid’ models.

Detail: Parametric Models

In addition to the geometric information, a parametric model has variables (parameters) that control its geometry. The shape and size of a model may be changed by modifying these parameters. Parametric models are built by constructing a sequence of geometric operations known as “Features”. Parametric

models are therefore ‘smart’ models that remember how they were built, and can ‘rethink’ themselves as their parameters or features are changed.

Detail: Non-parametric Models

A non-parametric model has none of the parameters or feature-driven geometry; it is sometimes referred to as a ‘dumb’ model. To modify a non-parametric model, it is often necessary to rebuild a lot of the geometry.

Detail: Hybrid Models

A Hybrid Model is neither completely parametric nor non-parametric; rather it is comprised of both types of geometry. Portions of the geometry may be so complicated that it is more efficient to make it non-parametric; while some of the geometry is still feature-driven. This is a very flexible and powerful modeling technique.

Building a completely parametric model takes more planning and foresight, however the extra effort can pay off if you are building multiple versions of a model, or if you can anticipate potential design modifications. The experienced NX5 user will determine which modeling approach is appropriate for each design project.

Detail: Curves

Curves can be created in three-dimensional space. Basic curves are typically non parametric, but NX5 has the capability to create some associative curves like lines, curves, circles, and helixes where you can control certain parameters and, unlike the basic curves, they appear in the modeling tree. Each curve has a set of control points associated with it. For example, lines have control points at both ends and at the midpoint, while circles have a basic control point at their center. These control points are useful for a variety of tasks, such as selecting features, building new geometry, or making measurements of existing geometry.

Curves can be placed in many ways, such as at x , y , z coordinates, or they can be connected to another curve at control points. They can also be placed in relation to other curves (parallel, perpendicular, tangent, normal, etc.).

Note: NX5 distinguishes between selecting a curve itself and selecting one of its control points. This can be a source of confusion to a new user. If you are having difficulty selecting geometry, pay close attention to your selection technique. To avoid the ambiguity between selecting a curve and its control points, you can zoom in closer to the area in question, or rotate the model into a better orientation. You can also use the snap point tool bar to control the selection.

Detail: Sketches

Sketches are by definition planar parametric features. Sketches are always defined upon a planar entity, with a reference that orients the sketch horizontally or vertically. As with curves, sketch entities also have control points.

The general approach in creating a sketch is to define its orientation, quickly create the curves that define it (do not worry about dimensions when creating geometry), and then to constrain it by placing dimensions or geometric constraints. At any point after the sketch is constrained, the dimensions that define the sketch can be changed and the model will then update.