NX 12 for Engineering Design

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FOREWORD

NX is one of the world's most advanced and tightly integrated CAD/CAM/CAE product development solution from Siemens PLM Software. Spanning the entire range of product development, NX delivers immense value to enterprises of all sizes. It simplifies complex product development, thus speeding up the process of introducing products to the market.

The NX software integrates multidisciplinary principles, conceptual design, 3D modeling, documentation, engineering analysis, graphic simulation, and concurrent engineering. The software has powerful hybrid modeling capabilities by integrating constraint-based feature modeling and explicit geometric modeling. In addition to modeling standard geometry parts, it allows the user to design complex freeform shapes such as airfoils and manifolds. It also merges solid and surface modeling techniques into one powerful toolset.

This self-guided tutorial provides a step-by-step approach for users to learn NX 12. It is intended for those with no previous experience with NX. However, users of previous versions of NX may also find this tutorial useful for them to learn the new user interfaces and functions. The user will be guided from starting an NX 12 session to creating models and designs that have various applications. Each chapter has components explained with the help of various dialog boxes and screenshots. These components are later used in the assembly modeling, machining and finite element analysis. The files of components are also available online to download and use. We first released the tutorial for Unigraphics 18 and later updated for NX 2 followed by the updates for NX 3, NX 5, NX 7, NX 9 and NX 10. This write-up further updates to NX 12.

If you have any questions or comments about this tutorial, please email Ming C. Leu at mleu@mst.edu or Wenjin Tao at wt6c2@mst.edu. The models and all the versions of the tutorial are available at http://web.mst.edu/~mleu.