Nx 9. 0 tutorials

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nx 9.0 Guide This homing tutorial provides a step-by-step approach for users to learn NX9.0. It is designed for those who have no previous experience with NX. However, users of previous versions of NX may also find this tutorial useful for them to learn new user interfaces and features. The user will be guided from the beginning of the NX9.0 session to create models and designs that have different applications. Each chapter has components explained through different dialog boxes and screen images. These components are later used in assembling, processing, and analyzing end elements. These component models are available online for download and use. First we released a tutorial for Unigraphics 18, then updated for NX2, followed by updates for NX3, NX5 and NX7. This is to write up to further update NX9.0. LINK DOWNLOAD (PDF) www.mediafire.com/download/rgcqfw6iibwta71/NX9.0+Manual.pdf incoming search terms:designmold nx9Siemens NX 9 0 Tutorials PDF concept. General Motors announced its DAC-1 system, and Bell Technologies introduced the GRAPHIC 1 remote display system. 4 NX 9.0 for engineering design by the University of Missouri Science and Technology during 1970-2019, the research efforts of the previous decade in computer graphics began to be fruitful, and the potential of interactive computer graphics in improving productivity was realized by industry, government and academia. The 1970-u2019s is characterized as a golden era for computer development and the launch of special instrumental design applications. The National Computer Graphics Association (NCGA) was established and the original graphics exchange specification (IGES) was launched. In 1980-1919, new theories and algorithms were developed, as well as the integration of various elements of design and production. The focus in research and development has been on expanding CAD/CAM systems beyond 3D geometric structures and providing more engineering applications. CURRENTLY, CAD/CAM is focused on effective and rapid integration and automation of various design and production elements, along with the development of new algorithms. There are many commercial CAD/CAM packages available for direct use that are comfortable and very experienced. Below are some of the commercial packages in the current market. uf0b7 AutoCAD and Mechanical Desktop are some low-level CAD software systems that are mainly used for 2D modeling and drawing. Uf0b7 NX, Pro-E, CATIA and I-DEAS are high-quality software modeling and design systems that are more expensive but more powerful. These software systems also have the capabilities of computer and engineering analysis. uf0b7 ANSYS, ABASUS, NATIONAL, Free and CFX are packages mainly used for analysis of structures and liquids. Different programs are used for different offers. For example, Fluent is used for liquids and used for structures. Uf0b7 Geomagic and CollabCAD are among the newest CAD systems that focus on collaborative design, allowing multiple software users to collaborate in the field of computer design over the Internet. 1.3 VIDEO/CAM/CAE Below are some of the terms used in this tutorial. 1.3.1 Computer Aid Design u2013 CAD CAD is a technology associated with the use of computer systems to assist in the creation, modification, analysis and optimization of design. Any computer program that embodies computer graphics and a program of applications that facilitate engineering functions during the design process can be classified as CAD software. 5 NX 9.0 for engineering design missouri University of Science and Technology The most important role of CAD is to determine the geometry of the design of the u2013 mechanical part, product assembly, architectural structure, electronic layout, layout of the building, etc. The greatest advantage of CAD systems is that they can save considerable time and reduce the number of errors caused otherwise by the need to redefine the geometry of the design from scratch every time it is necessary. 1.3.2 CAM CAM Computer Aided Manufacturing No2013 technology includes computer systems that plan, manage and control production operations through a computer interface with the plant's production resources. One of the most important areas of CAM is numerical control (NC). It is a method of using programmed instructions to control a machine that cuts, mills, grinds, punches or turns a raw stock into a finished part. Another important feature of CAM is robot programming. Process planning is also the goal of computer automation. 1.3.3 Computer Aided Engineering u2013 CAE technology uses a computer system to analyze the functions of a CAD product, allowing designers to model and learn how the product will behave so that the design can be refined and optimized. CAE tools are available for a variety of types of analyses. For example, kinematic analysis programs can be used to determine the pathways of movement and the speed of linking in the mechanisms. Dynamic analysis programs can be used to identify loads and shifts in complex assemblies such as cars. One of the most popular methods of analysis is the use of the End Element Method (FEM). This approach can be used to identify stress, deformation, heat transfer, magnetic field distribution, fluid flow and other continuous field problems that are often too complex to deal with any other approach. 1.4 SCOPE OF THIS TUTORIAL This tutorial is written for students and who are interested in learning how to use NX9.0 to design mechanical components and assemblies. Learning how to use this software will also be valuable for learning how to use other CAD systems such as PRO-E and CATIA. This tutorial provides a step-by-step approach for NX9.0. Topics include NX9.0, form features, operations functions, design, sketches, free form features, modeling and production. Chapter 1 provides a cad/CAM/CAE review. The product implementation cycle is discussed along with cad/CAM/CAE history and the definitions of each. Chapter 2 includes NX9.0 essentials from the start of the Windows session to the NX9.0 layout, practicing basic features such as printing, saving and exiting. It also provides a brief description of the coordinate system, layers, different tool sets, and other important commands that will be used in later chapters. 6 NX 9.0 for Engineering Design missouri University of Science and Technology Actual Design and Modeling Parts begins with Chapter 3. It describes various functions, such as reference functions, functions, and primitive functions, as well as how these features are used to create designs. Chapter 4 is an extension of Chapter 3, where different types of operations functions are performed on objects. Different types of operations include Trim, Blend, Boolean Operations and more. You'll learn how to create a drawing from a part model in Chapter 5. In this chapter, we demonstrate how to create a picture by adding views, measuring the drawings of the part and changing the different attributes of the drawing, such as text size, arrow size, and tolerance. Chapter 6 presents the concept of sketches. It describes how to sketch and give geometric and dimensional limitations. This chapter is very important, as modern components are very complex in geometry and difficult to model only with basic features. Chapter 7 introduces free-form modeling. The method of modeling curves and smooth surfaces will be demonstrated. Chapter 8 teaches the concepts of the Assembly's modelling and terminology. It describes top-down modeling and Bottom-Up modeling. We will use Bottom-Up modeling to build components into the product. Chapter 9 is phased in a brief introduction to the structure module available in NX9.0 to model and analyze the final elements. Chapter 10 will have a real-time experience of introducing an developed model into a production environment for processing. This chapter is dedicated to generating, testing, and modeling path tools to create CNC (Computer numerical codes) to produce developed parts from vertical processing centers. The examples and problems of exercise used in each chapter are designed to be one. NX 9.0 for Engineering Design Ming C Leu Albin Thomas Krishna Colan Department of Mechanical Engineering and Aerospace Engineering 2. FOREWORD Content 1 CHAPTER 1 -

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234 7. 1 NX 9.0 for engineering design missouri University of Science and Technology FOREWORD NX is one of the most advanced and tightly integrated CAD/CAM/CAE product development solutions. Covering the full range of product development, NX provides tremendous value for businesses of all sizes. It simplifies the complex design of products, thereby accelerating the process of introducing products to the market. NX software integrates principles based on knowledge, industrial design, geometric modeling, advanced analysis, graphic modeling and parallel design. The software has powerful hybrid modeling capabilities by integrating limited function modeling and explicit geometric modeling. In addition to modeling standard geometry parts, this allows the user to develop complex forms of free form such as gunpowder and variety. It also combines solid and superficial modeling techniques into one powerful toolkit. This self-correcting tutorial provides a step-by-step approach for users to learn NX9.0. It is designed for those who do not have NX experience. However, users of previous versions of NX may also find this tutorial useful for them to learn new user interfaces and features. The user will be guided from the beginning of the NX9.0 session to create models and designs that have different applications. Each chapter has components explained through different dialog boxes and screen images. These components are later used in assembling, processing, and analyzing end elements. These models and details available online for download and use. First we released a tutorial for Unigraphics 18, then updated for NX2, followed by updates for NX3, NX5 and NX7. This is to write up to further update NX9.0. Our previous efforts to prepare the NX self-guided tutorial have been funded by the National Science Foundation Advanced Technology Education Program and Partners for collaborative engineering education (PACE) program If you have any questions or comments on this tutorial, please email Min C. Leu on mleu@mst.edu or Albin Thomas in atgk3@mst.edu. Models and all versions of the tutorial are available on mleu/. 8. 2 NX 9.0 for Engineering Design Missouri University Science and Technology Chapter 1 - THE NEW Modern Production Environment can be characterized by a paradigm of product delivery of increasing diversity, fewer batches and higher quality in the context of growing global competition. The industry will not be able to survive worldwide competition if they do not introduce new products with better quality, lower costs and shorter management time. There is intense international competition and reduced availability of skilled labour. With dramatic changes in computing power and greater availability of software tools for design and manufacturing, engineers are now using Computer Aid Design (CAD), Computer Aid Manufacturing (CAM) and Computer Aid Engineering (CAE) systems to automate their design and manufacturing processes. These technologies are now used every day for a variety of engineering tasks. Here's a rundown of how CAD, CAM, and CAE technologies are used in the product implementation process. 1.1 PRODUCT REALIZE PROCESS The product implementation process can be roughly divided into two phases; design and production. The design process begins by identifying new customer needs and design variables that need to be improved, which are identified by marketing staff after receiving feedback from customers. Once the relevant design information is collected, the design specifications are formulated. A feasibility study is carried out with relevant project information and detailed design and analysis is carried out. Detailed design includes design conceptualization, promising product drawings, sketches and geometric modeling. The analysis includes stress analysis, jamming, kinematic analysis, mass property calculations and tolerance analysis, as well as design optimization. The quality of the results of these activities is directly related to the quality of the analysis and the tools used for the analysis. The manufacturing process begins with starting with production planning, which uses the drawings of the design process and finishes with the actual product. Process planning includes activities such as planning, material procurement and machine selection. There are various tasks, such as purchasing new tools, re programming and quality checks at different stages of the production process. Process planning involves planning all the processes used in the production of the product. Parts that pass quality checks are collected functionally tested, packaged, labeled and shipped to customers. Below is a diagram representing the product implementation process (Mastering CAD/CAM, Ibrahim Seid, McGraw Hill, 2005). 9. 3 NX 9.0 for Engineering Design Missouri University Science and Technology 1.2 CAD AND ENGINEERING DESIGN Computer Aid Design (CAD) technology has been in use since the beginning of the computer era. Ivan Sutherland developed a sketch system that demonstrated the ability to create drawings and quarrels of objects interactively on a cathode beam tube (CRT). The term CAD began to appear with the word design going beyond basic design concepts. General Motors announced its DAC-1 system, and Bell Technologies introduced the GRAPHIC 1 remote display system. 10. 4 NX 9.0 for engineering design by the University of Missouri Science and Technology in the 1970s, the research efforts of the previous decade in computer graphics began to be fruitful, and the potential of interactive computer graphics in improving productivity was realized by industry, government and academia. The 1970s are characterized as a golden era for computer development and the launch of special instrumental design applications. The National Computer Graphics Association (NCGA) was established and the original graphics exchange specification (IGES) was launched. In the 1980s, new theories and algorithms were developed and various elements of design and production were integrated. The focus in research and development has been on expanding CAD/CAM systems beyond 3D geometric structures and providing more engineering applications. In the CAD/CAM development focuses on efficient and rapid integration and automation of various elements design and production along with the development of new algorithms. There are many commercial CAD/CAM packages available for direct use that are comfortable and very experienced. Below are some of the commercial packages of interest, and the capabilities of each software. 11. 5 NX 9.0 for engineering design missouri University of Science and Technology The most basic role of CAD is to determine the geometry of design - mechanical part, product assembly, architectural structure, electronic layout, layout, etc. 1.3.2 Computer Aid Manufacturing - CAM CAM technology includes computer systems that plan, manage and control manufacturing operations through a computer interface with the factory's manufacturing resources. One of the most important areas of CAM is numerical control (NC). It is a method of using programmed instructions to control a machine that cuts, mill, grinds, punches or turns a raw stock into a finished part. Another important feature of CAM is robot programming. Process planning is also the goal of computer automation. 1.3.3 Computer Engineering - CAE CAE technology uses a computer system to analyze the functions of a CAD-created product that allows designers to explore how the product will behave so that the design can be refined most optimized. CAE tools are available for a variety of types of analyses. For example, kinematic analysis programmes can be used to determine the pathways of movement and speed in the mechanisms. Dynamic analysis programs can be used to identify loads and shifts in complex assemblies such as cars. One of the most popular methods of analysis is the use of the End Element Method (FEM). This approach can be used to identify stress, deformation, heat transfer, magnetic field distribution, fluid flow and other continuous field problems that are often too complex to deal with any other approach. 1.4 SCOPE OF THIS TUTORIAL This tutorial is written for students and engineers who are interested in learning how to use NX9.0 to design mechanical components and assemblies. Learning how to use this software will also be valuable for learning how to use other CAD systems such as PRO-E and CATIA. This tutorial provides a step-by-step approach for NX9.0 learning. Topics include starting with NX9.0, form functions, function operations, design, sketches, free form features, modeling and production. Chapter 1 provides a cad/CAM/CAE review. The product implementation cycle is discussed along with cad/CAM/CAE history and the definitions of each. Chapter 2 includes NX9.0 essentials from the start of the Windows session to the NX9.0 layout, practicing basic features such as printing, saving and exiting. It also provides a brief description of the coordinate system, layers, different tool sets, and other important commands that will be used in later chapters. 12. 6 NX 9.0 for Engineering Design missouri University of Science and Technology Actual Design and Simulation Parts begins with Chapter 3. It describes various functions, such as reference functions, functions, and primitive functions, as well as how these features are used to create designs. Chapter 4 is an extension of Chapter 3, where different types of operations functions are performed on objects. Different types of operations include Trim, Blend, Boolean Operations and more. You'll learn how to create a drawing from a part model in Chapter 5. In this chapter, we demonstrate how to create a picture by adding views, measuring the drawings of the part and changing the different attributes of the drawing, such as text size, arrow size, and tolerance. Chapter 6 presents the concept of sketches. It describes how to sketch and give geometric and dimensional limitations. This chapter is very important, as modern components are very complex in geometry and difficult to model only with basic features. Chapter 7 introduces free-form modeling. The method of modeling curves and smooth surfaces will be demonstrated. Chapter 8 teaches the concepts of the Assembly modeling and terminology. It describes top-down modeling and Bottom-Up modeling. We will use Bottom-Up modeling to build components in the product. Chapter 9 in a brief introduction to the structure the module is available in NX9.0 for for Modeling and analyzing the end elements. Chapter 10 will have a real-time experience of introducing an developed model into a production environment for processing. This chapter is dedicated to generating, testing, and modeling path tools to create the machine. Computer numerical control (CNC) machine implementation and production. Chapter 10 will also introduce the basic knowledge of CNC machine. 13. 7 NX 9.0 for Engineering Design Missouri University Science and Technology Chapter 1 - GETTING STARTED Previous versions of Unigraphics NX provide the basics required to use any CAD/CAM package. You'll learn preliminary steps to start, understand and use the NX9.0 package for modeling, development, etc. It contains five subcharraes a) Opening session NX9.0, b) Printing, saving and closing parts files, c) familiarity with the NX9.0 d) Use of layers and e) Understanding important commands and dialogues. 2.1 STARTING NX9.0 SESSION AND OPENING FILES 2.1.1 Open session NX9.0 - From the Windows desktop screen click on Start → All programs → Siemens NX 9.0 → NX 9.0 14. 8 NX 9.0 for Engineering Design Missouri University Science and Technology Basic NX9.0 Screen will open. This is the gateway to NX9.0 software. The blank screen of NX9.0 looks like the picture shown below. There will be various tips displayed on the screen about the features of the current version. The gateway also has a tape bar that will allow you to create a new file or open an existing file (under the home tab). On the left side of the Gateway screen is a toolbar called Resource Bar, which contains menus associated with different modules, and the ability to identify and modify the role of software, view the history of software use, and so on. This will be detailed later in this chapter. Let's start with learning how to open the file part in NX9.0. There are two options to create a new file. You can click on the new command at the top of the screen or go through the file drop down the tab. 2.1.2 Open new file - On the tape bar, under the file tab found in the top left corner of the screen, click FILE ➡ NEW This will open a new session, asking for the name and location of the new file to be created as shown in the bottom left corner. You need to select units (inches or millimeters) from working 15. 9 NX 9.0 for engineering design at the University of Missouri's Science and Technology Environment by clicking on the drop down menu in the top right corner. The default is millimetres. However, most of the material in the textbooks is modeled in inches. So always, be sure to pick inches before creating a new file Specified. You can also choose the type of file you want to create - either a part file or an assembly file or a sheet metal file - by selecting the type of file shown in the Templates dialog field in the center of the dialog box. The properties of the selected file are displayed below the preview in the middle right corner. Enter the location of the file and then click OK 2.1.3 Open the file part Click FILE → OPEN 16. 10 NX 9.0 for engineering design at The University of Missouri Science and Technology You can also click on the open icon from the Home tab at the top of the screen. Open Part File dialogue will appear. You can see a preview of the files on the right side of the window. You can disable the preview by pressing the box in front of the Preview button. - Click CANCEL to get out of window 2.2 PRINTING, SAVING AND CLOSING PART FILES 2.2.1 Print NX9.0 Image - Click FILE → PRINT The next figure shows the Print Dialog Window. Here you can choose a printer to use or specify the number of copies to be printed, the size of the paper and so on. You can also choose a scale for all three dimensions. You can also choose the printing method, i.e. wired frame, solid model by clicking on the right drop down menu. Clicking on the OK button will start the printing process. - Click CANCEL to get out of the window 2.2 Save part of the files It is important to save all the models created. It is also important to save all the models created in a specific folder. Create a folder in the root directory of the hard drive. Save all the models created in this folder. Open the file you want to save from the screen with the same name as before when creating the part file. - SAVE WORK PART ONLY: The option will only keep the active part on the screen. SAVE AS: The option allows you to save a piece on the screen using a different name. 18. 12 NX 9.0 for engineering design at the University of Missouri Science and Technology and save everything: This option will save all open file parts with their existing names. BOOKMARK: This option will keep a screenshot of the current model on the screen as. JPEG file and bookmarks. Remember, as in previous versions, all parts are saved with the .prt extension in NX9.0. 2.2.3 Close some of the files You can close the parts that are visible on the screen, click FILE → CLOSE If you close the file, the file will be cleared of working memory and any changes that are not saved will be lost. So don't forget to choose SAVE AND CLOSE or SAVE ALL AND CLOSE. For the first two options, selected parts or all parts of the files will be closed, but the NX9.0 session continues to work. 19. 13 NX 9.0 for Missouri Engineering Design Science and Technology 2.2.4 Exit NX9.0 Session - Click FILE → EXIT - Since we're not ready to get out of NX9.0, click NO If you have files open and made changes to them without saving, the message will ask you if you really want to quit. Choose NO, save the files, and then quit 2.2.5 while simultaneously saving all the pieces and leaving the second way of exiting the NX9.0 session at the same time save all the files and exit the program is - Click FILE → CLOSE → SAVE ALL and EXIT Save and Exit Alert Dialogue window shown below. Choose NO or CANCEL 20. 14 NX 9.0 for engineering design at the University of Missouri Science and Technology 2.3 NX9.0 INTERFACE The NX9.0 user interface is very simple with the help of various icons. Most commands can be performed by navigating the mouse across the screen and clicking on the icons. Keyboard entries are mostly limited to entering values and file names. 2.3.1 Mouse Functionality is highly recommended to use a three-cap mouse or scrolling mouse while working with NX9.0. The strength of the mouse buttons and their main features are discussed below. 2.3.1.1 Left Mouse Button (MB1): The MB1 button or left mouse is used to select icons, menus, and other entities on the graphics screen. Double-clicking MB1 on any feature will automatically open the Edit Dialog window. 2.3.1.2 Average mouse button (MB2): The MB2 button or the middle mouse or scroll button is used to rotate the object by pressing, holding and dragging. It can be used for Pan and Zoom in conjunction with other mouse buttons or key buttons. If its a scroll button, you can zoom in and scroll. Simply clicking or MB2 will perform an OK command if any pop-up or dialog is open. 2.3.1.3 Right Mouse Button (MB3): MB3 or right mouse button is used to access the pop-up user menu. You can access the following options that pop up depending on the selection mode and the app. The numbers on the right are in the sketch application. When you click on MB3, you'll select options related to this feature (Object/Action Menu): Clicking on the MB3 and holding the button will display a set of icons around the mouse pointer near the right edge of the graphics screen and turn. Similarly, for the vertical axis and axis perpendicular to the screen, click on the bottom edge and top edge of the screen respectively and turn. If you keep clicking on the MB2 in the same position for a few seconds, it will fix the rotation point (an orange circle symbol appears) and you can drag around the object to view. Zoom in /Out: Tap and hold both the left mouse button and the middle button (or scroll button) at the same time and drag OR Click and hold the button on the keyboard, and then press and drag the middle mouse button. Or scroll up and down if the mouse has a scroll wheel. 22. 16 NX 9.0 for engineering design at the University of Missouri Science and Technology Pan: - Press and hold both the middle button and the right mouse button at the same time and drag or press the button and hold the button on the keyboard and press and drag the middle button of the mouse. 2.3.2 NX9.0 Gateway The next digit shows a typical layout of the NX9.0 window when the file is open. This is the NX9.0 gateway from where you can choose any module to work with, such as modeling, manufacturing, etc. Toolbars can be placed anywhere or position on the screen. Look for the same set of icons. Below is the NX9.0 program in the simulation app. 23. 17 NX 9.0 for engineering design by Missouri University of Science and Technology 2.3.2.1 Features gateway zone tape bar: Tape bar interface gives the user the ability to easily access different commands without reducing the graphic box area. Teams are organized in tape bars under different tabs and groups for easy recognition and accessibility. For example, in the tape bar is shown in the picture above, we have a home, a curve, the surface of tabs. In the home tab we have a direct sketch, function, synchronized modeling and superficial groups. And in each group we have a set of teams. Unlike previous versions of NX, this is a new feature developed in NX 9.0 overlooking touch systems. 24. 18 NX 9.0 for engineering design by the University of Missouri Science and Technology to dock back on the main toolbar, click down the arrow on the undocking tab located at the top right of the tab and press the dock tab. Choice Bar: Choice Bar is located below active toolbars and displays choices. These options include filters и точки привязки для<Shift> <Ctrl> <Ctrl><Ctrl> Functions. Resource Bar: Resource Bar features icons for a number of pages in one place, using very little UI space. NX9.0 places all the navigator windows in the resource bar, as well as a history palette, build navigator, Part navigator, animation navigator, simulation navigator, roles and web browser. By default, the resource bar is located on the left side of the NX9.0 window. You can dock and undock resource bars by clicking on the pin icon in the upper right corner of the resource bar. Undocking the resource bar will allow you to move it to the right side of the screen. Status Line: The status line shows the current status of the program. For example, when you are loading a large build, the status line will show the percentage of the completed operation. When the operation is complete, the system displays the next appropriate signal. 25. 19 NX 9.0 for Engineering Design Missouri University Science and Technology 2.3.2.2 Part Navigator - Click on the icon part of the navigator, the third icon from above on the resource bar Part Navigator provides a visual representation of parents and children's relationship functions in the working part in a separate window in tree type format. It shows all the primitives, entities used in modeling. This allows you to perform a variety of editing activities on these functions. For example, you can use Part Navigator to suppress or suppress objects or change their parameters or positioning sizes. Removing the green tick mark will suppress the function. The software will give a warning if the child's parental relationship is disrupted by suppressing any particular function. Part Navigator is available for all NX applications, not just modeling. However, you can only do function editing operations if you're in a simulation module. Editing the feature in Part Navigator will automatically update the model. Editing features will be discussed later. 2.3.2.3 Story - Click on the story icon, the seventh on top on The History Palette provides quick access to newly opened files or other palette records. It can be used to reboot parts that have been recently worked on or repeatedly add a small set of palette elements to the model. The story palette remembers the last variants of the palette that were used and the state of the session when it was closed. NX stores the palettes that have been uploaded to the session and restores them in the next session. The system does not clear the palette of history Part. 26. 20 NX 9.0 for Engineering Engineering University of Missouri Science and Technology to reuse the piece, drag it out of the history palette into the graphic box. To reboot the part, click on the session bookmark saved. 2.3.3 Geometry Geometry Choice Properties are very advanced in NX9.0. You can filter out a selection method that makes it easier to select geometry in a close cluster. In addition, you can perform any of the function options that NX9.0 reasonably provides depending on the entity you choose. The cursor of the mouse on the graphics screen is usually in the form of a crosshair, as shown in the picture. The choice of elements can be based on the degree of essence, like, the choice of geometric entities, features and components. You can choose the method by selecting one of the icons in the toolbar of choice. Feature selection: Clicking on any of the icons in the picture below will allow you to select the features in the part file. It will not select basic entities such as edges, faces, etc. Selected features can also be applied to a part or the entire build depending on the requirement. In addition, object filtering can be further narrowed by selecting one of the desired options in the drop-off menu, as shown in the picture below. For example, if you select CURVE, only the curves on the screen will be highlighted from the option. By default, THERE IS NO SELECTION FILTER. A common choice of objects: Tapping on the icon, as shown in the image below, will allow you to select the entire object in the graphics window. Tapping on the icon will highlight the entire object in the graphics window. 27. 21 NX 9.0 for engineering design at the University of Missouri Science and Technology Tap on the icon, as shown in the picture below, will highlight the geometry of the solid. If you want to select the edge of the solid, tap some of the hidden geometry projected onto the screen. After a couple of seconds, the ball cursor turns into a plus symbol, as shown in the picture. Click the left mouse button (MB1) to get the Confirmation of Choice dialog, as shown in the next digit below. This quickPick menu consists of a list of entities captured in the cursor ball. The entities are located in the order of increasing the degree of the entity. For example, lower numbers are assigned to edges and verticals, while higher numbers are given to solids. By moving the cursor to the numbers displayed, NX9.0 will highlight the corresponding essence on the screen in a purple color. For example, in the picture below, the person on top is assigned the number '5'. Similarly, hidden entities will also be highlighted with a number on the list. You can browse and click on a number that matches the desired object or function. 2.3.4 User Preferences 28. 22 NX 9.0 for Engineering Design missouri University of Science and Technology - PREFERENCES Select on the Menu button located at the top right of the main window to find the different options available to users' preferences used to determine the parameters of displaying new objects, names, layouts and views. You can set the layer, color, font, and width of the objects you create. You can also design layouts and views, control the display of the names and boundaries of objects and views, change the size of the selection ball, specify the method of the rectangle of choice, establish tolerance and chain method, and design and activate the grid. Changes made with the Preference menu redefine any defaults for the same features. User Interface - Choose preferences→USER INTERFACE to find options in the dialog field. The user interface customizes NX and interacts with the specifications you set. You can control the location, size and visibility of the main window, graphic display and information window. You can set the number of decimal places (accuracy) that the system uses for both input text fields and data displayed in the information window. You can also specify a full or small dialogue to select a file. You can also set macro parameters and include a confirmation dialogue for Unsubstanti operations. The overall tab allows you to set the level of accuracy as seen from the information window, the Layout tab allows you to establish the location of the resource bar, the Macro tab allows you to pause when displaying the visualization of the animation - Select PREFERENCES → VISUALIZATION to find options in the dialog window. This dialog controls attributes that affect the display in the graphics box. Some attributes are related to a part or to certain part views. The settings for these attributes are stored in the part file. For many of these attributes, when you create a new part or view, the setting is initiated to a value specified at 29. 23 NX 9.0 for engineering design of the University of Missouri Science and Technology customer default file. Other attributes are related to the session and apply to all parts of the session. Some of these attributes are saved from session to session in the registry. For some session attributes, the option can be initiated to the default value of the client' variable environment. Click on a different tab button to find the options in the dialog field. Click on EDIT BACKGROUND to get another dialog box. You can change your color scheme to the color that you want. The background color of the graphic window. Window: supports graded backgrounds for Shaded or Wireframe displays. The background can be the same or graded for all backgrounds color from 0 to 255. Click OK when you're done you can change the background of the graphics box using preferences. Choose PREFERENCES → BACKGROUND - You can also click ON THE → OBJECT This will be a pop-up of the OBJECT PREFERENCES dialogue or EDIT OBJECT DISPLAY. Change and observe the color and transparency of a solid object. This is not limited to solid objects. You can also apply this option to individual solid entities. For example, you can click on any specific hard surface and apply display settings. 30. 24 NX 9.0 for engineering design by The University of Missouri Science and Technology 2.3.5 Applications Unlike older versions of NX, APPLICATIONS can be opened using the file option located in the top left corner of the main window. You can choose the type of app you want to run from the fall down menu. For example, you can choose modeling, design, build, and so on, as shown. The default app that starts when you open a file or start a new file is a simulation. 2.4 COORDINATE SYSTEMS There are different coordinate systems in NX. The three-axis symbol is used to identify the coordinate system. 2.4.1 Absolute coordinate system 31. 25 NX 9.0 for engineering design of the University of Missouri Science and Technology Absolute coordinate system is the coordinate system from which all objects are referred. It is a fixed coordinate system and therefore the location and orientation of each object in the NX9.0 simulation space is tied back into that system. Absolute system coordination (or Absolute CSYS) also provides a common frame of reference between part files. The absolute position in the X-1, Y-1 and No.1 in the one-part file is also the same location in any other part file. View Triad is a visual indicator that is the orientation of the system absolute coordinates of the Model 2.4.2 System of Coordinates System (WCS) is something that you will use for construction when you want to determine the orientation and angles of functions. WCS axis is designated XC, YC, and WCS. (C means current). You can have multiple coordinate systems in the part file, but only one of them can be a work coordinate system. 2.4.4 Move WCS Here, you'll learn how to translate and rotate WCS. Choose MENU FORMAT WCS 32. 26 NX 9.0 for engineering design missouri University of Science and Technology 2.4.4.1 Translation WCS This procedure will move WCS origin at any time you specify, but the orientation (direction of axes) of WCS will remain the same. Choose FORMAT →WCS → ORIGIN Current Designer Dialogue Figure. You can point a point from the drop menu at the top of the dialog window or enter the X-Y-I coordinates in the XC, YC, and qC fields. Most of the work will be related to the system of coordinates of work, and not with the absolute system of coordinates. The default is WCS. The default action button is Inferred Point. The button is highlighted, as shown in the picture. The active icon name appears above the top row of the action buttons. This is the point on the object that is closest to the cursor. It can be any of the binding points, such as the center of the circle or the end point of the line and so on. Click CANCEL 2.4.4.2 Turn WCS You can also turn WCS around one of its axes. Select FORMAT → WCS → THE ROTATE Dialogue ROTATE WCS is displayed on the right side. The dialogue shows six different ways WCS rotates around the axis. These rotation procedures follow the right-hand rotation rule. You can also specify the angle at which WCS will rotate. Click CANCEL 2.4.4.3 Save your current WCS location and orientation to use as a permanent coordinate system. Select format → WCS → 33. 27 NX 9.0 for engineering design missouri University of Science and Technology 2.5 USING LAYERS Layers are used to store objects in a structured and consistent manner. Unlike simple visual tools such as folders and hide, Layers provides a permanent and work environment. Layers provides a permanent and work environment. Unlike simple visual tools such as folders and hide, Layers provides a permanent and work environment. Layers provides a permanent and work environment. Unlike simple visual tools such as folders and hide, Layers provides a permanent and work environment. 2.5.1 Control the layer with NX9.0, you can control whether objects are visible or selected by layers. A layer is a system attribute, such as color, font, and width, that all objects in NX9.0 must have. NX9.0 has 256 layers, one of which is always the working layer. Any of the 256 layers can be assigned to one of the four status classifications. The work you can choose from - visible only the invisible working layer - is the layer on which objects are created and which is always visible and selected as long as it remains the working layer. Layer 1 is the default work layer when you start a new part file. When the work layer is switched to another type of layer, the previous work layer automatically becomes selected, and then can be assigned the status of visible only or invisible. The number of objects that can be on one layer is unlimited. You have the freedom to choose the layer on which you want to create an object and the status of that layer. To assign status to layers or layers, select FORMAT → LAYER SETTINGS, however, it should be noted that using company layer standards would be beneficial to maintain consistency between files. 2.5.2 Teams in layers We will follow simple steps to practice commands in layers. First will create two 34. 28 NX 9.0 for engineering design by the University of Missouri Science and Technology Objects (solids) by method as follows. Details of solid modeling will be discussed in the next chapter. The solids that we draw here are just to practice in this chapter. Select the FILE → NEW Name file and select the folder in which to save it. Make sure you choose the units to be inches into the drop menu. Select file type as model - Click OK - Select MENU INSERT DESIGN FEATURE CYLINDER - Select AXIS, DIAMETER, HEIGHT under type - Click on the icon next to the Vector Report The Designer's Dialogue will appear. This is to indicate the direction of the cylinder axis. The default direction will be in the direction of q. Click OK on the pop-up - Click on the icon next to the window point the constructor will appear for you to determine the location of the cylinder. The default location will be the source (0.0.0) on WCS. Click OK If you want to change the direction of the axis of point of origin, click on the boxes laid out in red ink, as shown in the picture. The next type is 2 inches for diameter and 4 inches for height under the properties - Click OK - Click CANCEL on any other menu that appears. Now the screen will look like this. 35. 29 NX 9.0 for Engineering Design missouri University of Science and Technology - Right to tap the screen and select ORIENT VIEW → TRIMETRIC - If the solid is in the wired frame, right tap the screen and select RENDERING STYLE → SHADED OR click on the Shaded icon in the toolbar Now you will be able to see a solid cylinder. 36. 30 NX 9.0 for engineering design missouri University of Science and Technology Now let's practice some layer of command. Choose FORMAT → MOVE TO LAYER you will be asked to choose an object. Move the cursor to the cylinder and press it so that it becomes highlighted. Click OK You'll get the following pop-up. In the destination layer or space category at the top of the window, type 25. Choose APPLY, followed by CANCEL 37. 31 NX 9.0 for engineering design at the University of Missouri Science and Technology order has now gone to the 25th layer. It can no longer be seen in layer 1. To see the cylinder, click FORMAT → LAYER SETTINGS You can see that layer 25 has an object, while by default Working Layer 1 has no objects. Click OK Cylinder will again be visible on the screen. Save the file as we will use it later in the tutorial. 38. 32 NX 9.0 for engineering design at The University of Missouri Science and Technology 2.6 IMPORTANT COMMANDS/DIALOGS In this section you will learn some important commands and dialogues that will be useful in modeling and sketching. 2.6.1 Toolbars contain icons that serve as shortcuts to many NX9.0 features. Figure on on the right, the main elements of the toolbar are displayed, usually displayed. However, you can find many more icons for different function commands, based on the selected module and how the module is configured. The right click anywhere on the existing toolbars gives a list of other toolbars. You can add any of the toolbars by checking them. The list of toolbars that can be seen in the default option is standard, viewing, choice, utility, etc. If you want to add buttons related to commands not displayed, click on the toolbar and select the buttons. Click on the icon to add the button to the toolbar. Click on the icon to remove the button from the toolbar. Choose CUSTOMIZE from the toolbar to add or remove buttons from the toolbar. Click on the icon to reset the toolbar to the default settings. Click on the icon to dock the toolbar to the resource bar. 39. 34 NX 9.0 engineering design Missouri University science and technology Raleigh tab has a variety of toolbar menu settings that are displayed on the NX9.0 interface. This allows you to set up the toolbars you want to display in the interface. 2.6.2 Function conversion is a powerful feature for scaling, mirroring, and changing the existing part. Click on MENU EDIT TRANSFORM Here, we have to choose an entity such as a solid or curves or sketch. You can choose one feature or more by clicking on the features. 41. 35 NX 9.0 for engineering design missouri University of Science and Technology - Click on the cylinder so that it gets stressed - Click OK This opens a dialogue box that allows you to perform many functions like zoom, and mirror the image of the part of the model as shown in the next digit. You can choose any of these menus. The scale scales the model. You can create a copy of an object using a mirror across a line or mirror across a plane, create multiple copies of the essence, such as a curve, surface or solid using a rectangular and circular array. If you want to move an object to a fixed entity, you can use the Move menu - Click on edit → MOVE OBJECT 42. 36 NX 9.0 for engineering design missouri University science and technology dialogue field opens with options, as shown in the picture above. Notice the handle of the object that appears on the object as a coordinate system. You can choose the type of movement from the motion drop-down menu. Default With this, you can move the object in any direction. There are several other ways object, as shown in the picture below. POINT TO POINT - This option allows you to move the center of the cylinder to any destination on the X-Y-I axis you want to move. The coordinates are based on WCS. 43. 37 NX 9.0 for engineering design of The University of Missouri Science and Technology - DISTANCE - This option moves the selected object in the direction of X-Y-I to the distance you enter. Click on SPECIFY VECTOR Select Direction. Type 5 in the DISTANCE box. This will translate the cylinder to a distance of 5 inches along the X-Axis - Click OK Note that MOVE ORIGINAL is RESULT clicked. The cylinder will move in the direction of X at a distance of 5 inches. Click CANCEL As you can see, we have moved the cylinder in the direction of X. Similarly, we can also copy the cylinder at a given distance or to a specified location by selecting the COPY ORIGINAL option in RESULT. These are the main teams that you need initially. This completes the implementation of the NX9.0 interface framework and some of the core functions of operations that can be done. In the next chapter, we learn more about the shape features and some primitive types of objects. 44. 38 NX 9.0 for Engineering Design Missouri University of Science and Technology Chapter 3 - FORM FEATURES This chapter will discuss the basics of Form Features in NX9.0: Essentially, we will discuss what function, what different types of functions, what primitives and how to simulate features in NX9.0 using primitives. This will start the NX9.0 part model and develop an understanding of the use of form functions for modeling. In NX9.0, features are classified into three basic form features: primitive, function and function operations. Primitives are the basic building blocks of the model. Primitives are created by the function of the function. Function operations are made by the function of the function. Function operations are made by the function of the function. 45. 39 NX 9.0 for Engineering Design Missouri University Science and Technology 3.2 TYPE FEATURES There are six types of feature form: commonalities include blocks, cylinders, cones, spheres, extruded bodies, and rotating bodies. Usually functions can be classified as the following bodies: a class of objects containing solids and sheets. Solid: A collection of faces and edges that enclose volume. Body leaf: a collection of one or more individuals who do not attach volume. Face: The area on the outside of the body, fenced with edges. 46. 39 NX 9.0 for Engineering Design Missouri University Science and Technology 3.2 TYPE FEATURES Reference Features, Swept Features, Delete Features, User-defined Features, Extract and primitives. Just like the NX7 version, the NX9.0 version stores all form features under the INSERT menu option. Form features are also available in the Form Features toolbar. Click INSERT on the Menu button As you can see, the marked menus on the picture on the right contain commands of form functions. Icons of the form function are grouped into the home tool panel, as shown below. You can choose the icons that you use often. Click on the drop arrow in the home toolbar - Select feature features of the FEATURE GROUP link that allow you to create reference planes or reference axes. These links can help you create features on cylinders, cones, spheres and rotating solids. 46. 40 NX 9.0 for Engineering Design missouri University of Science and Technology - Click on MENU INSERT DATUM/POINT to view various reference features: Datum Plane, Datum Axis, Datum CSYS, and Point Swept Features They allow you to create a body by extruding or rotating sketch geometry. Swept Features include: Extruded Body' Rotating Body Sweeping Along the Guide Stylized sweep to choose a rideable feature you can do the following: Click on INSERT → DESIGN FEATURE for Extrude and Revolve or 47. 41 NX 9.0 for Engineering Design at Missouri University of Science and Technology - Click on INSERT → SWEEP for the rest of the Options Remove Features Remove Features allow you to create a body by removing a solid part from other parts. Click on THE INSERT → DESIGN FUNCTION Delete Features include, Hole Boss Pocket and Pad Slot Groove You can also choose features by clicking on the custom features icons They allow you to create your own form features to automate widely used design elements. User-defined features can expand the range and power of built-in form functions. Click on INSERT → DESIGN FEATURE → USER DEFINED

CHAMFER - Choose two inner edges as shown punished, and press OK - Enter the diameter of Chamfer Offset as 0.0436 inches and press OK You will see a daisy on the nut. Save the model. 86. 80 NX 9.0 for engineering design of the University of Missouri Science and Technology 4.3.4 Model rack with instances In this part, we will practice to create instances of this object. Open file Arborpress_rack.prt - Select INSERT → DESIGN FEATURE → POCKET - Select RECTANGULAR in the pop-up - Click on the top of the rack as shown in the picture for the placement surface. Keep an eye on the Cue line. He gives the following instructions. 87. 81 NX 9.0 for Engineering Design missouri University of Science and Technology - Click on the Edge as shown in the image for horizontal reference This will be a pop-up of parameters. Enter the settings values as shown in the picture and select OK - Select The Wireframe option in Display mode for greater clarity - When the positioning window pops up, select PERPENDICULAR 88. 82 NX 9.0 for Engineering Design missouri University of Science and Technology - Click on the Edge on The Hard, and then click on the blue dotted line as shown below - Enter the expression value as 37.8 and select OK - once again choose the PERPENDICULAR option and then choose another set of edges along the Y-axis as shown in the picture below. NOTE: Choose the blue line along the Y axis (perpendicular to the last selected blue line) 89. 83 NX 9.0 for Engineering Design missouri University of Science and Technology - Enter the meaning of expression as 10 and choose OK twice. Choose CANCEL. Now the model will look like this. Let's create instances of the slot, like the teeth of the rack, which will cling to the pinion. Select INSERT → ASSOCIATIVE COPY→ PATTERN FEATURE - Click on the choice feature. Now click on the rectangular pocket you've created. Select the layout as linear - Point the vector as a positive XC direction - Select COUNT AND PITCH's Option and type value to count as 19, and that to step distance like 9.4 - Choose OK - Click YES - Click CANCEL Model rack will look like shown. Figure.

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