

# **NX DESIGN ASSOCIATE CERTIFICATION |2025 Update |Complete Exam Set Questions And Verified Answers (solutions) | Get It 100% Accurate!!| Already Passed A+**

Which command could you use to control the display of a specific feature based on the value of an expression? - ✓✓ANSWER✓✓>>Suppress by Expression

What type of feature is used to define a new Sketch on a non-planar face? - ✓✓ANSWER✓✓>>Datum Plane

How can you investigate the design intent of a sketch? - ✓✓ANSWER✓✓>>Vary a dimension to see how the sketch changes

What command allows you to create a feature by rotating a section string around an axis through a specified angle? - ✓✓ANSWER✓✓>>revolve

To create a datum plane at an angle to a face, what would you select? - ✓✓ANSWER✓✓>>Select a face and datum axis or straight edge

Which of the following best describes an advantage of creating an edge blend feature instead of creating fillet curves inside a sketch? - ✓✓ANSWER✓✓>>Can be edited, suppressed, or deleted without affecting the sketch

Which one of these are not a type of hole you can create? - ✓✓ANSWER✓✓>>external threads

Which of the following cannot be selected to create Edge Blend features? - ✓✓ANSWER✓✓>>sketch curves

What type of chamfer cannot be created? - ✓✓ANSWER✓✓>>two angle

What is an advantage of using a Hole feature instead of an Extrude? - ✓✓ANSWER✓✓>>A Hole feature can be easily edited to change the form

You create a threaded hole in a lug for a specific size fastener. The resulting Hole feature includes: - ✓✓ANSWER✓✓>>A single feature The hole, threads, and the entry and exit chamfers

Which of the following conditions must be in place before you can reattach a sketch to new geometry? - ✓✓ANSWER✓✓>>The geometry must have a timestamp before the sketch

When you use Edit Parameters or Edit with Rollback, you generally access ... - ✓✓ANSWER✓✓>>the same dialog box used to create the feature

How can you easily edit a feature of a model? - ✓✓ANSWER✓✓>>Double click the feature in the part navigator

Which one of the following feature types can you browse in the Browser? - ✓✓ANSWER✓✓>>Body-based features

What does the feature browser do? - ✓✓ANSWER✓✓>>in a separate window, explores parent-child relations between features and their ancestor and descendant objects

When using Edit with Rollback .. - ✓✓ANSWER✓✓>>the Feature being edited is made the current feature, all later features are suppressed, once the edit is complete the model is rebuilt to the end

To display other solutions for a Linear dimension you may use .. - ✓✓ANSWER✓✓>>the Alternate Solution command or Enter a negative value for the dimension type

What does a warning symbol on a feature in the part navigator mean? - ✓✓ANSWER✓✓>>Serious problem - the feature has failed to update

If you use the Shell command on a solid body and do not select at least one face to pierce which of the following will the result be? - ✓✓ANSWER✓✓>>A hollow solid body

When using the Trim Body command, the vector that is displayed points.. - ✓✓ANSWER✓✓>>toward the portion of the body that will be removed

When you use the Shell command, the wall thickness vector points in the \_\_\_\_\_ - ✓✓ANSWER✓✓>>positive direction of wall thickness

When creating Offset Curves, for the Distance, Draft and Law Control type of offset curves, the curves to offset.. - ✓✓ANSWER✓✓>>must lie in the same plane

Draftcommand is typically used to apply \_\_\_\_\_ to faces for use in molded or die cast parts. - ✓✓ANSWER✓✓>>slope

While creating a shell, you need to remove a face that is adjacent to a blend face. What setting needs to be set to help prevent failures? - ✓✓ANSWER✓✓>>add shelf face at tangent edge

How do you create offset curves in both directions at the same time? - ✓✓ANSWER✓✓>>Toggle on Symmetric Offset

A characteristic of a model that is required for a molded or cast model typically includes - ✓✓ANSWER✓✓>>A model with drafted sides

When designing a molded part that needs to be released from two halves of a mold, what should be designed into the part? - ✓✓ANSWER✓✓>>a parting line

What option will create a body with wall thickness from an extrude feature? - ✓✓ANSWER✓✓>>offset

Which of one the following statements is false when using the Shell command? - ✓✓ANSWER✓✓>>When creating multiple alternate thicknesses, each are created as separate shell feature

Which one of the following statements is generally true for molded part designs? - ✓✓ANSWER✓✓>>Molded parts are designed with thin walls to promote thermal cooling in the mold

When using the Sweep Along Guide command, which one of these can you not do? - ✓✓ANSWER✓✓>>Select multiple close-looped sections

When using the Sweep Along Guide command what two sets of curves must be defined? - ✓✓ANSWER✓✓>>Section & Guide

The Pattern Feature and Pattern Face commands produce similar results in most cases. If the faces you select don't bind a region that can be added or subtracted to the solid body, which command should you use? - ✓✓ANSWER✓✓>>Pattern Geometry

What pattern would you use to create a pattern on an imported step file? - ✓✓ANSWER✓✓>>Pattern Face

What pattern method allows you to pick from and to points? - ✓✓ANSWER✓✓>>General

What is a unique property of Pattern Geometry? - ✓✓ANSWER✓✓>>It does not attempt to perform a Boolean of the results

The Mirror Feature command creates a new feature. When you edit this feature which of the following can you do? - ✓✓ANSWER✓✓>>Select a different symmetry plane

What command will allow you to quickly create the other half of a symmetric part model? - ✓✓ANSWER✓✓>>Extract Geometry - Mirror Body

You want to add a symmetry constraint between pairs of curves already created in your sketch. Which command would you use? - ✓✓ANSWER✓✓>>Make Symmetric

Which command would you use to apply taper or slope to faces, bodies, or from edges? - ✓✓ANSWER✓✓>>draft

Why would you use a separate Thread command? - ✓✓ANSWER✓✓>>To create external threads

You can create a chamfer feature or chamfer curves inside a sketch. What is a potential advantage of creating chamfers in a sketch? - ✓✓ANSWER✓✓>>Your model history will have fewer features

Why would you want to deactivate the Timestamp Order option in the Part Navigator? - ✓✓ANSWER✓✓>>To view solid bodies in the model

When selecting a feature in the Part Navigator, what do the colors of the text above and below the selected Feature mean? - ✓✓ANSWER✓✓>>The Parent- and Child relationships

When selecting a feature in the Part Navigator what color does the text of its children features then become? - ✓✓ANSWER✓✓>>blue

In the Part Navigator how can you list the parameters, expressions, and values of a feature without using the Show Dimensions command? - ✓✓ANSWER✓✓>>Open the Details Group

After using the Show Dimensions command on a sketch how do you erase the temporarily-displayed dimensions from the graphics window? - ✓✓ANSWER✓✓>>Use the Refresh command

By default, what happens to the child features when their parent feature is deleted? - ✓✓ANSWER✓✓>>The child features are automatically deleted

Which Synchronous Modeling command can be used to change the radii of this feature regardless of the feature history? - ✓✓ANSWER✓✓>>Resize Blend