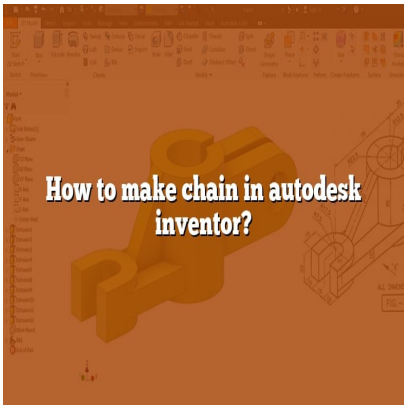


How to make chain in autodesk inventor?

Description

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With this article you will have the answer to your How to make chain in autodesk inventor? question. Indeed Autodesk inventor is even easier if you have access to the free Autodesk inventor tutorials and the answers to questions like. Our CAD-Elearning.com site contains all the articles that will help you progress in the study of this wonderful software. Browse through our site and you will find different articles answering your different questions.
The use of parametric design in Autodesk inventor makes it a powerful tool for designers and engineers. The designer can observe the impact of adjustments on neighboring components or even on the entire solution. This makes it quick and easy for designers to identify and solve problems.
And here is the answer to your How to make chain in autodesk inventor? question, read on.

Introduction

1. Select the chain midplane for the **chain** drive coordinate system.
2. To select a **chain**, click Browse.
3. Within the browse window, select the chain type and the size.
4. Add 2 sprockets.
5. Specify the position of each sprocket by dragging a grip at the center of the sprocket.

You asked, how do you use the chain dimension in **Inventor**?

1. On the ribbon, click Annotate tab Dimension panel Chain . Alternatively, on the ribbon, click Annotate tab Dimension panel Chain Set .
2. Select the geometry to dimension in the graphics window:
3. Right-click and select Continue.
4. Click to place the dimensions.
5. Right-click and select Create.

Beside above, how do you design a **chain**?

Considering this, how do you link parts in Inventor?

Additionally, how do I create a keychain in Inventor? Chain Dimensioning is when dimensions are drawn from one feature to a second feature, then a third feature is located relative to the second feature. The chain can continue with any number of features, with each one being located in relation to the preceding feature.

How do you show dimensions in Inventor drawing?

Right-click a feature in the browser or graphics window and choose Show Dimensions. Right-click a sketch in the browser and toggle on Dimension Visibility. Select View Tab, Visibility panel, Object Visibility, and select or deselect the Sketch Dimensions check box to show or hide all sketch dimensions.

How do you design a chain drive?

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1. Calculate the drive ratio R (velocity ratio) given the input RPM and output RPM.
2. Select sprocket tooth numbers.
3. Determine the Service (selection) factor.
4. Calculate the Design (selection) power.

What is chain in machine design?

Chain drives are flexible and made of number of links and its a intermediate between belts and gears drives. Chains can only be used to transmit power between parallel shafts. Unlike belt drives, chain drives uses special toothed wheels called sprockets.

What are the types of chain drive?

1. Power transmission chain drive.
2. Conveyor chain drive.
3. Hoisting and hauling chain drive.

What is iLogic inventor?

iLogic is a functionality of Inventor that allows users and administrators to create logic in the form of VB.net to accomplish tasks. Rules are developed and organized using snippets and other code writing statements to run at given times to consistently do some of the work engineers and designers need to do.

What is an iPart inventor?

An iPart is a table driven master part that configures standard parts to different sizes and states. The table can be edited within Inventor or externally in a spreadsheet. Each row can control feature state (enabled or suppressed), and many other variables such as feature size, appearance, material, and part number.

How do you parameterize in Inventor?

1. At the bottom of the Parameters dialog, do one of the following:
2. Click in the Parameter Name box to enter the name.
3. Edit the Equation box to set the parameterâ€™s value.
4. Select Key if the parameter is a key parameter.

How big is a keychain hole?

The Keyring hole is 5mm thick or . 19 inches.

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How big should a keychain be?

For best results we suggest keeping the design as simple as possible. Less is more when personalizing your own keychains. Many customers forget that the average keychain size is only 1â€³X2â€³ in size.

How do you make a name tag in Inventor?

What is a chain dimension?

Chain dimensions are reference dimensions driven by the geometry. You cannot change their values or use the values to drive the model. When you create chain dimensions, your first selection defines the starting edge of the chain. Subsequent selections are measured from one selection to the next.

What is the problem with chain dimensioning?

The obvious case for chain dimensions is when tolerances arenâ€™t important, such as a rough cut, or a series of outside dimensions with loose tolerances. The other is a case where the dimension from edge isnâ€™t as critical as another dimension.

How do you draw parallel dimensions?

To draw a parallel dimension line, click the Parallel dimension tool in the toolbox. To draw a vertical or horizontal dimension line, click the Horizontal or vertical dimension tool . 2 . Click to place the starting point, and drag to where you want to place the endpoint of the dimension line.

How do you create a dual dimension in Inventor drawing?

Expand the Dimensions area. Highlight an existing Dimensions style on the Right and select â€œNewâ€•. Give it a â€œtestâ€• name for test. Select ok.

How do I create a sectional view in Inventor?

1. Select an existing view to use as the parent view.
2. On the ribbon, click Place Views tab Create panel Section .
3. Select the view cutting line you defined in the sketch.
4. (Optional) Specify the appearance of the view in the Section View dialog box.

Bottom line:

I sincerely hope that this article has provided you with all of the How to make chain in autodesk inventor? information that you require. If you have any further queries regarding Autodesk inventor software, please explore our CAD-Elearning.com site, where you will discover various Autodesk inventor tutorials answers. Thank you for your time. If this isnâ€™t the case, please donâ€™t be hesitant about letting me know in the comments below or on the contact page.

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The article provides clarification on the following points:

- How do you design a chain drive?
- What is chain in machine design?
- What are the types of chain drive?
- How do you parameterize in Inventor?
- How big is a keychain hole?
- How do you make a name tag in Inventor?
- What is a chain dimension?
- How do you draw parallel dimensions?
- How do you create a dual dimension in Inventor drawing?
- How do I create a sectional view in Inventor?